Creo 2.0
Curriculum Guide
Live Classroom Curriculum Guide

- Update to Creo Parametric 2.0 from Creo Elements/Pro 5.0
- Update to Creo Parametric 2.0 from Pro/ENGINEER Wildfire 4.0
- Introduction to Creo Parametric 2.0
- Advanced Modeling using Creo Parametric 2.0
- Advanced Assembly Design using Creo Parametric 2.0
- Detailing using Creo Parametric 2.0
- Surfacing using Creo Parametric 2.0
- Sheetmetal Design using Creo Parametric 2.0
- Milling using Creo Parametric 2.0
- Cabling using Creo Parametric 2.0
- Piping using Creo Parametric 2.0
- Introduction to Creo Simulate 2.0
- Introduction to Creo Schematics 2.0
- Introduction to Creo Layout 2.0
- Introduction to Creo Illustrate 2.0
- Flexible Modeling using Creo Parametric 2.0
- Introduction to Creo Direct 2.0
- Introduction to Creo Options Modeler 2.0
- Mold Design using Creo Parametric 2.0
- Interactive Surface Design using Creo Parametric 2.0
- Behavioral Modeling using Creo Parametric 2.0
- Mechanism Design using Creo Parametric 2.0
- Mechanism Simulation using Creo Parametric 2.0
- Turning using Creo Parametric 2.0
- Manufacturing Update for Creo Parametric 2.0 from Creo Elements/Pro 5.0
- Advanced Turning and Multi-task Machining using Creo Parametric 2.0
- Creo Parametric 2.0 for SolidWorks Users
Update to Creo Parametric 2.0 from Creo Elements/Pro 5.0

Overview

<table>
<thead>
<tr>
<th>Course Code</th>
<th>TRN-3900-T</th>
</tr>
</thead>
<tbody>
<tr>
<td>Course Length</td>
<td>1 Day</td>
</tr>
</tbody>
</table>

In this course, you will learn how to utilize the core functionality enhancements in Creo Parametric 2.0. First, you will become familiar with using and customizing the new ribbon interface in Creo Parametric. The new measure and sectioning interfaces will also be examined. Next, you will become familiar with the Sketcher workflow and reference enhancements. Part modeling enhancements to features such as Extrude, Corner Chamfer, Sweeps, Blends, and Datum Curves will then be examined. You will also learn about new and enhanced Assembly capabilities, such as selecting multiple components, the new relationship constraints, and enhancements for dragging components. Then, you will examine the new Table and Balloon functionality for 2-D drawings, and review various detailing enhancements. Finally, in Sheetmetal mode you will learn to use the many updated Wall, Bend, and Relief tools, as well as the consolidated Flat Pattern tool and configuring Sheetmetal properties.

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

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

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

Audience

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

### Day 1

<table>
<thead>
<tr>
<th>Module</th>
<th>1</th>
<th>Interface Enhancements</th>
</tr>
</thead>
<tbody>
<tr>
<td>Module</td>
<td>2</td>
<td>Sketcher Enhancements</td>
</tr>
<tr>
<td>Module</td>
<td>3</td>
<td>Part Modeling Enhancements</td>
</tr>
<tr>
<td>Module</td>
<td>4</td>
<td>Assembly Enhancements</td>
</tr>
<tr>
<td>Module</td>
<td>5</td>
<td>Drawing Enhancements</td>
</tr>
<tr>
<td>Module</td>
<td>6</td>
<td>Sheetmetal Enhancements</td>
</tr>
<tr>
<td>Module</td>
<td>7</td>
<td>Appendix</td>
</tr>
</tbody>
</table>
Course Content

Module 1. Interface Enhancements
   i. Understanding the Main Interface
   ii. Creo Graphics Enhancements
   iii. Understanding the Ribbon Interface
   iv. Customizing the Ribbon Interface
   v. Configuring Creo Options
   vi. Selection Enhancements
   vii. Locate in Model Tree Enhancement
   viii. Using the Measure Tools
   ix. Using the Measure Summary Tool
   x. Creating Planar Part Cross-Sections
   xi. Creating Assembly Cross-Sections
   xii. Anti-Alias Edge Display Enhancement

Knowledge Check Questions

Module 2. Sketcher Enhancements
   i. Sketcher Interface Enhancements
   ii. Sketcher Workflow Enhancements
   iii. Sketching Rectangles and Parallelograms

Knowledge Check Questions

Module 3. Part Modeling Enhancements
   i. Feature Preview Enhancements
   ii. Understanding Regeneration and Auto Regeneration
   iii. Editing Features
   iv. Automatically Adding and Removing Material
   v. Adding Taper to Extrude Features
   vi. Creating Corner Chamfers
   vii. Variable Section and Simple Sweep Consolidation
   viii. Creating Sweeps with Open Trajectories
   ix. Creating Sweeps with Closed Trajectories
   x. Analyzing Sweep Feature Attributes
   xi. Helical Sweep Enhancements
   xii. Creating Helical Sweeps for Springs
   xiii. Analyzing Helical Sweep Profile and Pitch Variations
   xiv. Utilizing Variable Sections in Helical Sweeps
   xv. Creating Blends by Selecting Parallel Sections
   xvi. Creating Blends by Selecting Non-Parallel Sections
   xvii. Creating Blends by Sketching Sections
   xviii. Analyzing Blend Section Tools
   xix. Analyzing Blend Options
Module 4. Assembly Enhancements
   i. Reference Dialog Enhancements
   ii. Assembly Unit and Mass Properties Enhancements
   iii. Component Rename Enhancements
   iv. Assembly Appearance Enhancements
   v. Assembly Model Tree Enhancements
   vi. Selecting Multiple Components
   vii. Orienting Components
   viii. Creating Coincident Constraints using Geometry
   ix. Creating Coincident Constraints using Datum Features
   x. Creating Distance Constraints
   xi. Creating Parallel, Normal, and Angle Constraints
   xii. Assembling using Automatic
   xiii. Retrieving Assembly Subsets
   xiv. Lightweight Graphics Representations
   xv. Restructuring and Reordering Assembly Components
   xvi. Configuring Constraint Sets with Parameters
   xvii. Reviewing Assembly Changes

Module 5. Drawing Enhancements
   i. Detail Options Enhancements
   ii. Detailing Ribbon and Selection Enhancements
   iii. Detailing Graphics Enhancements
   iv. Prefix and Suffix Enhancements
   v. Feature-Specific Text Enhancements
   vi. Detailing Arrow Enhancements
   vii. Detail and Section View Enhancements

Knowledge Check Questions
viii. Dimension Text Orientation Enhancements  
ix. View Scale Enhancements  
x. BOM Enhancements  
xi. Inserting Tables  
xii. Editing Table Properties  
xiii. Creating Tables from File  
xiv. Creating BOM Balloons  
xv. Dimension and Annotation Enhancements  
xvi. Creating Combination States and Showing Annotations  
xvii. Creating Annotation Planes and Annotation Features  
xviii. Managing Annotation Display

Knowledge Check Questions

Module 6. Sheetmetal Enhancements

i. Sheetmetal Properties Enhancements
ii. Miscellaneous Sheetmetal Enhancements
iii. Extending and Trimming Walls
iv. Creating Offset Walls
v. Creating Bend Features
vi. Adding Transition to Bends
vii. Creating Planar Bends
viii. Creating Edge Bends
ix. Creating Unbend Features
x. Creating Bend Back Features
xi. Joining Walls
xii. Creating Sketched Forms
xiii. Flattening Forms and Unstamping Edges
xiv. Creating Split Area Features
xv. Creating Corner Relief
xvi. Creating Rip Features
xvii. Converting Solid Models to Sheetmetal
xviii. Using Conversion Features
xix. Previewing and Creating Flat Patterns
xx. Creating Flat States

Knowledge Check Questions

Module 7. Appendix

Knowledge Check Questions
Update to Creo Parametric 2.0 from Pro/ENGINEER Wildfire 4.0

Overview

<table>
<thead>
<tr>
<th>Course Code</th>
<th>TRN-3901-T</th>
</tr>
</thead>
<tbody>
<tr>
<td>Course Length</td>
<td>2 Days</td>
</tr>
</tbody>
</table>

In this course, you will learn how to utilize the core functionality enhancements in Creo Parametric 2.0. First, you will become familiar with using and customizing the new ribbon interface in Creo Parametric. The new measure and sectioning interfaces will also be examined. Next, you will study the Sketcher workflow and reference enhancements, as well as Sketcher constraint, geometry, and diagnostics enhancements. Part modeling enhancements to features such as Extrude, Trajectory Rib, Point Pattern, Corner Chamfer, Sweeps, Blends, UDFs, and Datum Curves will then be examined. You will also learn about new and enhanced Assembly capabilities such as selecting multiple components, the new relationship constraints, enhancements for dragging components, explode enhancements, simplified rep enhancements, and dynamic gear enhancements. Then you will learn how to identify and resolve part and assembly failures on the fly, without entering Resolve mode. Next, you will examine the new Drawing tree and drawing sheets tabs, as well as showing annotations. Also in Drawing mode, you will learn the new Table and Balloon functionality for 2-D drawings, and review various detailing enhancements. Finally, in Sheetmetal mode you will learn to use the updated Wall, Bend, and Relief tools as well as the consolidated Flat Pattern tool and configuring Sheetmetal properties.

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

- Utilize the Interface enhancements in Creo Parametric 2.0
- Utilize the Sketcher enhancements in Creo Parametric 2.0
- Utilize the Modeling enhancements in Creo Parametric 2.0
- Utilize the Assembly enhancements in Creo Parametric 2.0
- Utilize the Drawing enhancements in Creo Parametric 2.0
- Utilize the Sheetmetal enhancements in Creo Parametric 2.0

Prerequisites

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

Audience

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

## Day 1

<table>
<thead>
<tr>
<th>Module</th>
<th>1</th>
<th>Interface Enhancements</th>
</tr>
</thead>
<tbody>
<tr>
<td>Module</td>
<td>2</td>
<td>Sketcher Enhancements</td>
</tr>
<tr>
<td>Module</td>
<td>3</td>
<td>Part Modeling Enhancements</td>
</tr>
<tr>
<td>Module</td>
<td>4</td>
<td>Assembly Enhancements</td>
</tr>
</tbody>
</table>

## Day 2

<table>
<thead>
<tr>
<th>Module</th>
<th>5</th>
<th>Resolving Failures</th>
</tr>
</thead>
<tbody>
<tr>
<td>Module</td>
<td>6</td>
<td>Drawing Enhancements</td>
</tr>
<tr>
<td>Module</td>
<td>7</td>
<td>Sheetmetal Enhancements</td>
</tr>
<tr>
<td>Module</td>
<td>8</td>
<td>Appendix</td>
</tr>
</tbody>
</table>
Course Content

Module 1. Interface Enhancements
   i. Understanding the Main Interface
   ii. Creo Graphics Enhancements
   iii. Understanding the Ribbon Interface
   iv. Customizing the Ribbon Interface
   v. Configuring Creo Options
   vi. Selection Enhancements
   vii. Insert Position Enhancements
   viii. Creating Layer States
   ix. Managing and Editing Appearances
   x. Locate in Model Tree Enhancement
   xi. Using the Measure Tools
   xii. Using the Measure Summary Tool
   xiii. Creating Planar Part Cross-Sections
   xiv. Creating Assembly Cross-Sections
   xv. Anti-Alias Edge Display Enhancement

Knowledge Check Questions

Module 2. Sketcher Enhancements
   i. Sketcher Interface Enhancements
   ii. Sketcher Workflow Enhancements
   iii. Utilizing Constraints
   iv. Sketching with On-the-Fly Constraints
   v. Sketcher Dimension Enhancements
   vi. Locking Sketcher Entities
   vii. Sketching Rectangles and Parallelograms
   viii. Sketching Chamfers
   ix. Sketching Ellipses
   x. Thickening Edges
   xi. Sketching Geometry Datums
   xii. Sketcher Diagnostic Tools

Knowledge Check Questions

Module 3. Part Modeling Enhancements
   i. Lightweight Hole Display
   ii. Intersect at Surface Round Transition
   iii. Creating Curvature Continuous Rounds
   iv. Feature Preview Enhancements
   v. Understanding Regeneration and Auto Regeneration
   vi. Editing Features
   vii. Automatically Adding and Removing Material
viii. Adding Taper to Extrude Features  
ix. Creating Corner Chamfers  
x. Creating Trajectory Ribs  
xi. Variable Section and Simple Sweep Consolidation  
xii. Creating Sweeps with Open Trajectories  
xiii. Creating Sweeps with Closed Trajectories  
xiv. Analyzing Sweep Feature Attributes  
 xv. Helical Sweep Enhancements  
 xvi. Creating Helical Sweeps for Springs  
xvii. Analyzing Helical Sweep Profile and Pitch Variations  
xviii. Utilizing Variable Sections in Helical Sweeps  
xix. Creating Blends by Selecting Parallel Sections  
xx. Creating Blends by Selecting Non-Parallel Sections  
xxi. Creating Blends by Sketching Sections  
xxii. Analyzing Blend Section Tools  
xxiii. Analyzing Blend Options  
xxiv. Analyzing Blend Tangency  
xxv. Creating Rotational Blends by Selecting Sections  
xxvi. Creating Rotational Blends by Sketching Sections  
xxvii. Analyzing Rotational Blend Options  
xxviii. Analyzing Rotational Blend Tangency  
xxix. Creating Geometry Patterns  
xxx. Creating Point Patterns  
xxxi. Direction Patterning with Multiple Direction Types  
xxii. Creating Axes from Coordinate Systems  
xxiii. Creating On-Surface Coordinate Systems  
xxiv. UDF Enhancements  
xxv. Creating UDFs Using On-Surface Coordinate Systems  
xxvi. Creating a Curve From Equation  
xxvii. Creating a Curve from a Cross-Section  
xxviii. Creating Curves Through a Point or Vertex  
xxix. Creating a Curve Through a Point Array  
  x. Creating Cosmetic Sketches  
  xii. Creating Cosmetic Threads  
  xiii. Using the Draft Analysis Option  

Knowledge Check Questions

Module 4. Assembly Enhancements
  i. Reference Dialog Enhancements  
  ii. Assembly Unit and Mass Properties Enhancements  
  iii. Component Rename Enhancements  
  iv. Generic Instance Enhancements
Module 5. Resolving Failures

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. Understanding Resolve Mode Tools
ix. Recovering Models
Module 6. Drawing Enhancements

- i. Understanding the Drawing Ribbon User Interface
- ii. Exploring Drawing Ribbon Commands
- iii. Detailing Ribbon and Selection Enhancements
- iv. Utilizing the Drawing Tree
- v. Managing Drawing Sheets
- vi. Showing, Erasing, and Deleting Annotations
- vii. Dimension Enhancements
- viii. Detailing Enhancements
- ix. Printing Enhancements
- x. Detail Options Enhancements
- xi. Detailing Graphics Enhancements
- xii. Prefix and Suffix Enhancements
- xiii. Feature-Specific Text Enhancements
- xiv. Detailing Arrow Enhancements
- xv. Detail and Section View Enhancements
- xvi. Dimension Text Orientation Enhancements
- xvii. View Scale Enhancements
- xviii. BOM Enhancements
- xix. Inserting Tables
- xx. Editing Table Properties
- xxi. Creating Tables from File
- xxii. Creating BOM Balloons
- xxiii. Dimension and Annotation Enhancements
- xxiv. Creating Combination States and Showing Annotations
- xxv. Creating Annotation Planes and Annotation Features
- xxvi. Managing Annotation Display

Knowledge Check Questions

Module 7. Sheetmetal Enhancements

- i. Sheetmetal Properties Enhancements
- ii. Miscellaneous Sheetmetal Enhancements
- iii. Extending and Trimming Walls
- iv. Creating Offset Walls
- v. Creating Bend Features
- vi. Adding Transition to Bends
- vii. Creating Planar Bends
- viii. Creating Edge Bends
- ix. Creating Unbend Features

Knowledge Check Questions
x. Creating Bend Back Features
xi. Joining Walls
xii. Creating Sketched Forms
xiii. Flattening Forms and Unstamping Edges
xiv. Creating Split Area Features
xv. Creating Corner Relief
xvi. Creating Rip Features
xvii. Converting Solid Models to Sheetmetal
xviii. Using Conversion Features
xix. Previewing and Creating Flat Patterns
xx. Creating Flat States
xxi. Punch Form Enhancements
xxii. Utilizing Punch Model Annotations
xxiii. Flatten Form Enhancements
xxiv. Patterning Walls
xxv. Mirroring Walls
xxvi. Csys Follow Surface Option

Knowledge Check Questions

Module 8. Appendix

Knowledge Check Questions
Introduction to Creo Parametric 2.0

Overview

Course Code  TRN-3902-T  
Course Length  5 Days

In this course, you will learn core modeling skills and quickly become proficient with Creo Parametric 2.0. 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 2.0.

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

Course Objectives

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

Prerequisites

• None

Audience

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

## Day 1

<table>
<thead>
<tr>
<th>Module</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Introduction to the Creo Parametric Basic Modeling Process</td>
</tr>
<tr>
<td>2</td>
<td>Understanding Creo Parametric Concepts</td>
</tr>
<tr>
<td>3</td>
<td>Using the Creo Parametric Interface</td>
</tr>
<tr>
<td>4</td>
<td>Selecting Geometry, Features, and Models</td>
</tr>
<tr>
<td>5</td>
<td>Editing Geometry, Features, and Models</td>
</tr>
<tr>
<td>6</td>
<td>Creating Sketcher Geometry</td>
</tr>
</tbody>
</table>

## Day 2

<table>
<thead>
<tr>
<th>Module</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>7</td>
<td>Using Sketcher Tools</td>
</tr>
<tr>
<td>8</td>
<td>Creating Sketches for Features</td>
</tr>
<tr>
<td>9</td>
<td>Creating Datum Features: Planes and Axes</td>
</tr>
<tr>
<td>10</td>
<td>Creating Extrudes, Revolves, and Ribs</td>
</tr>
<tr>
<td>11</td>
<td>Utilizing Internal Sketches and Embedded Datums</td>
</tr>
<tr>
<td>12</td>
<td>Creating Sweeps and Blends</td>
</tr>
</tbody>
</table>

## Day 3

<table>
<thead>
<tr>
<th>Module</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>13</td>
<td>Creating Holes, Shells, and Draft</td>
</tr>
<tr>
<td>14</td>
<td>Creating Rounds and Chamfers</td>
</tr>
<tr>
<td>15</td>
<td>Project I</td>
</tr>
<tr>
<td>16</td>
<td>Group, Copy, and Mirror Tools</td>
</tr>
<tr>
<td>17</td>
<td>Creating Patterns</td>
</tr>
<tr>
<td>18</td>
<td>Measuring and Inspecting Models</td>
</tr>
</tbody>
</table>

## Day 4

<table>
<thead>
<tr>
<th>Module</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>19</td>
<td>Assembling with Constraints</td>
</tr>
<tr>
<td>20</td>
<td>Assembling with Connections</td>
</tr>
<tr>
<td>21</td>
<td>Exploding Assemblies</td>
</tr>
<tr>
<td>22</td>
<td>Drawing Layout and Views</td>
</tr>
<tr>
<td>23</td>
<td>Creating Drawing Annotations</td>
</tr>
<tr>
<td>24</td>
<td>Using Layers</td>
</tr>
</tbody>
</table>
## Day 5

<table>
<thead>
<tr>
<th>Module</th>
<th>25</th>
<th>Investigating Parent/Child Relationships</th>
</tr>
</thead>
<tbody>
<tr>
<td>Module</td>
<td>26</td>
<td>Capturing and Managing Design Intent</td>
</tr>
<tr>
<td>Module</td>
<td>27</td>
<td>Resolving Failures and Seeking Help</td>
</tr>
<tr>
<td>Module</td>
<td>28</td>
<td>Project II</td>
</tr>
</tbody>
</table>
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. Customizing the Ribbon Interface
   vii. Working with Multiple Windows
   viii. Managing Files in Creo Parametric
   ix. Understanding Datum Display Options
   x. Understanding Display Style Options
   xi. Analyzing Basic 3-D Orientation
   xii. Understanding the View Manager
   xiii. Creating and Managing View Orientations
   xiv. Creating Style States Using the View Manager
   xv. Managing and Editing Appearances
   xvi. 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. Using Keyboard Shortcuts
   iv. Understanding the Model Tree
   v. Understanding Model Tree Filters
   vi. Understanding Basic Model Tree Columns
   vii. Selecting Items using Direct Selection
   viii. Selecting Items using Query Selection
   ix. Using the Search Tool
x. Using the Smart Selection Filter
xi. Understanding Selection Filters
xii. Selecting Multiple Components

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
v. Thickening Edges

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. Automatically Adding and Removing Material
ix. Creating Profile Rib Features

Knowledge Check Questions

Module 11. Utilizing Internal Sketches and Embedded Datums
i. Creating Internal Sketches
ii. 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
ix. Analyzing Draft Hinges and Pull Direction

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. Creating New Drawings and Applying Formats
iv. Creating and Orienting General Views  
v. Utilizing the Drawing Tree  
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 New Drawings using Drawing Templates  
xiii. Modifying Drawing Views  
xiv. Creating Assembly and Exploded Views

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. Creating Layer States  
v. 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. Understanding Resolve Mode Tools
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
Advanced Modeling using Creo Parametric 2.0

Overview

Course Code  TRN-3903-T
Course Length  3 Days

The Advanced Modeling using Creo Parametric 2.0 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 2.0.

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 2.0
- Update to Creo Parametric 2.0 from Creo Elements/Pro 5.0

Audience

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

Day 1

<table>
<thead>
<tr>
<th>Module</th>
<th>Module</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Advanced Selection</td>
</tr>
<tr>
<td>2</td>
<td>Advanced Datum Features</td>
</tr>
<tr>
<td>3</td>
<td>Advanced Sketching</td>
</tr>
<tr>
<td>4</td>
<td>Advanced Hole Creation</td>
</tr>
<tr>
<td>5</td>
<td>Advanced Drafts and Ribs</td>
</tr>
<tr>
<td>6</td>
<td>Advanced Shells</td>
</tr>
<tr>
<td>7</td>
<td>Advanced Rounds and Chamfers</td>
</tr>
</tbody>
</table>

Day 2

<table>
<thead>
<tr>
<th>Module</th>
<th>Module</th>
</tr>
</thead>
<tbody>
<tr>
<td>8</td>
<td>Relations and Parameters</td>
</tr>
<tr>
<td>9</td>
<td>Advanced Blends</td>
</tr>
<tr>
<td>10</td>
<td>Sweeps with Variable Sections</td>
</tr>
<tr>
<td>11</td>
<td>Helical Sweeps</td>
</tr>
<tr>
<td>12</td>
<td>Swept Blends</td>
</tr>
</tbody>
</table>

Day 3

<table>
<thead>
<tr>
<th>Module</th>
<th>Module</th>
</tr>
</thead>
<tbody>
<tr>
<td>13</td>
<td>Advanced Layers</td>
</tr>
<tr>
<td>14</td>
<td>Advanced Reference Management</td>
</tr>
<tr>
<td>15</td>
<td>Family Tables</td>
</tr>
<tr>
<td>16</td>
<td>Reusing Features</td>
</tr>
<tr>
<td>17</td>
<td>Advanced Copy</td>
</tr>
<tr>
<td>18</td>
<td>Advanced Patterns</td>
</tr>
</tbody>
</table>
Course Content

Module 1. Advanced Selection
   i. Advanced Chain Selection
   ii. Advanced Surface Selection

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. Analyzing Sketcher Convert Options
   xi. Locking Sketcher Entities
   xii. Analyzing Sketcher Dimension Options
   xiii. 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. Creating Drafts with Multiple Angles
   iii. Using the Extend Intersect Surfaces Draft Option
   iv. Creating Drafts Split at Sketch
   v. Creating Drafts Split at Curve
   vi. Creating Drafts Split at Surface
   vii. Creating Drafts with Variable Pull Direction
   viii. 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. Analyzing Additional Chamfer Types
   xi. Analyzing Advanced Chamfer Dimensioning Schemes
   xii. Analyzing Chamfer Creation Methods
   xiii. Creating Corner Chamfers
   xiv. Creating Chamfers by Reference
   xv. Analyzing Chamfer References and Pieces
   xvi. Using Intent Edges for Chamfers
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
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

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

Knowledge Check Questions

Module 13. Advanced Layers
i. Understanding Layers
ii. Creating and Managing Layers
iii. Creating Layer Rules
iv. 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
v. Editing Family Table Members

Knowledge Check Questions

Module 16. Reusing Features
i. Creating UDFs
ii. Placing UDFs
iii. Creating UDFs Using On-Surface Coordinate Systems
iv. Creating Inheritance Features
v. Using External Merge to Add Material
vi. Using External Merge to Remove Material

Knowledge Check Questions

Module 17. Advanced Copy
i. Configuring Independency
ii. Analyzing Advanced Reference Configuration
iii. Copying Features Fully Dependent with Options to Vary

Knowledge Check Questions

Module 18. Advanced Patterns
i. Understanding Pattern Regeneration Options
ii. Creating Dimension Patterns in One Direction
iii. Creating Dimension Patterns in Two Directions
iv. Creating Rotational Dimension Patterns
v. Creating Geometry Patterns
vi. Creating Fill Patterns
vii. Specifying Fill Pattern Settings
viii. Creating Pattern Tables
ix. Applying Pattern Tables
x. Creating Curve Patterns
xi. Creating Point Patterns
xii. Unpatterning Group Patterns
xiii. Creating Patterns of Patterns
xiv. Moving/Mirroring Patterns

Knowledge Check Questions
Overview

Course Code | TRN-3904-T
Course Length | 3 Days

In this course, you will learn how to use Creo Parametric 2.0 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 assembly constraints
- Create and using component interfaces
- Create and using 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, layer states, and combined views
- Substitute components using user-defined reps, envelopes, and simplified reps
- Understand advanced simplified rep functionality
- Create and use assembly structure and skeletons
Prerequisites

• Introduction to Creo Parametric 2.0
• Update to Creo Parametric 2.0 from Creo Elements/Pro 5.0

Audience

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

### Day 1

<table>
<thead>
<tr>
<th>Module</th>
<th>1</th>
<th>Using Advanced Assembly Constraints</th>
</tr>
</thead>
<tbody>
<tr>
<td>Module</td>
<td>2</td>
<td>Creating and Using Component Interfaces</td>
</tr>
<tr>
<td>Module</td>
<td>3</td>
<td>Creating and Using Flexible Components</td>
</tr>
<tr>
<td>Module</td>
<td>4</td>
<td>Restructuring and Mirroring Assemblies</td>
</tr>
</tbody>
</table>

### Day 2

<table>
<thead>
<tr>
<th>Module</th>
<th>5</th>
<th>Using Assembly Features and Shrinkwrap</th>
</tr>
</thead>
<tbody>
<tr>
<td>Module</td>
<td>6</td>
<td>Replacing Components in an Assembly</td>
</tr>
<tr>
<td>Module</td>
<td>7</td>
<td>Understanding the Basics of Simplified Reps</td>
</tr>
<tr>
<td>Module</td>
<td>8</td>
<td>Creating Cross-Sections, Display Styles, Layer States, and Combined Views</td>
</tr>
</tbody>
</table>

### Day 3

<table>
<thead>
<tr>
<th>Module</th>
<th>9</th>
<th>Substituting Components using User Defined, Envelopes, and Simplified Reps</th>
</tr>
</thead>
<tbody>
<tr>
<td>Module</td>
<td>10</td>
<td>Understanding Advanced Simplified Rep Functionality</td>
</tr>
<tr>
<td>Module</td>
<td>11</td>
<td>Creating and Using Assembly Structure and Skeletons</td>
</tr>
<tr>
<td>Module</td>
<td>12</td>
<td>Project</td>
</tr>
</tbody>
</table>


Course Content

Module 1. Using Advanced Assembly Constraints
   i. Constraining Components using Fix
   ii. Constraining Two Coordinate Systems using Coincident
   iii. Constraining a Point on a Line using Coincident
   iv. Constraining a Point on a Surface using Coincident
   v. Constraining an Edge on a Surface using Coincident
   vi. Constraining a Point on a Point using Coincident
   vii. Creating a Tangent Constraint using Auto Constrain
   viii. Configuring Constraint Sets with Parameters

Knowledge Check Questions

Module 2. 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 3. 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 4. 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 5. 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 6. 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 7. 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 8. Creating Cross-Sections, Display Styles, Layer States, and Combined Views
i. Understanding Assembly Cross-Sections
ii. Creating Assembly Cross-Sections
iii. Creating Offset Assembly Cross-Sections
iv. Creating Zone Assembly Cross-Sections
v. Creating Display Styles
vi. Creating Layer States in an Assembly
vii. Creating Combination Views

Knowledge Check Questions

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
Module 10. 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 11. 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 12. 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
Detailing using Creo Parametric 2.0

Overview

<table>
<thead>
<tr>
<th>Course Code</th>
<th>TRN-3905-T</th>
</tr>
</thead>
<tbody>
<tr>
<td>Course Length</td>
<td>3 Days</td>
</tr>
</tbody>
</table>

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 2.0 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 2.0

Audience

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

## Day 1

<table>
<thead>
<tr>
<th>Module</th>
<th>1</th>
<th>Introduction to Drawings</th>
</tr>
</thead>
<tbody>
<tr>
<td>Module</td>
<td>2</td>
<td>Creating New Drawings</td>
</tr>
<tr>
<td>Module</td>
<td>3</td>
<td>Creating Drawing Views</td>
</tr>
</tbody>
</table>

## Day 2

<table>
<thead>
<tr>
<th>Module</th>
<th>4</th>
<th>Adding Model Details to Drawings</th>
</tr>
</thead>
<tbody>
<tr>
<td>Module</td>
<td>5</td>
<td>Adding Notes to Drawings</td>
</tr>
<tr>
<td>Module</td>
<td>6</td>
<td>Adding Tolerance Information</td>
</tr>
<tr>
<td>Module</td>
<td>7</td>
<td>Adding Draft Geometry and Symbols</td>
</tr>
</tbody>
</table>

## Day 3

<table>
<thead>
<tr>
<th>Module</th>
<th>8</th>
<th>Using Layers in Drawings</th>
</tr>
</thead>
<tbody>
<tr>
<td>Module</td>
<td>9</td>
<td>Creating and Using Tables in Drawings</td>
</tr>
<tr>
<td>Module</td>
<td>10</td>
<td>Using Report Information in Drawings</td>
</tr>
<tr>
<td>Module</td>
<td>11</td>
<td>Creating Drawing Formats</td>
</tr>
<tr>
<td>Module</td>
<td>12</td>
<td>Configuring the Drawing Environment</td>
</tr>
<tr>
<td>Module</td>
<td>13</td>
<td>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

Knowledge Check Questions

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

Knowledge Check Questions

Module 3. Creating Drawing Views
   i. Configuring Drawing Models
   ii. Configuring Drawing Sheets
   iii. Adding General Views
   iv. Adding Projection Views
   v. Editing Drawing Views
   vi. Editing Visible View Area
   vii. Adding Detailed Views
   viii. Adding Auxiliary Views
   ix. Understanding Cross-Section Concepts and View Types
   x. Adding 2-D Cross-Section Views
   xi. Modifying Cross Hatching Display
   xii. Adding Assembly Exploded Views

Knowledge Check Questions

Module 4. Adding Model Details to Drawings
   i. Understanding Annotations in Drawings
   ii. Showing, Erasing, and Deleting Annotations
   iii. Inserting Driven Dimensions
   iv. Inserting Ordinate Dimensions
   v. Adjusting Dimensions and Detail Items
   vi. Changing Dimension Display

Knowledge Check Questions

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

Knowledge Check Questions

Module 6. Adding Tolerance Information
i. Understanding Dimensional Tolerances
ii. Configuring Dimensional Tolerances
iii. Understanding Geometric Tolerances
iv. Setting Up Geometric Tolerance References
v. Applying Geometric Tolerances

Knowledge Check Questions

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

Knowledge Check Questions

Module 8. Using Layers in Drawings
i. Understanding Layers in Drawings
ii. Using Layers in Drawings

Knowledge Check Questions

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

Knowledge Check Questions

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

Knowledge Check Questions

Module 11. Creating Drawing Formats
i. Creating Drawing Formats

Knowledge Check Questions

Module 12. Configuring the Drawing Environment
i. Configuring the Drawing Environment

Module 13. Managing Large Drawings
i. Understanding Drawing Regeneration
ii. Managing Large Drawings

Knowledge Check Questions
Surfacing using Creo Parametric 2.0

Overview

Course Code: TRN-3906-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 2.0
- Update to Creo Parametric 2.0 from Creo Elements/Pro 5.0

Audience

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

### Day 1

<table>
<thead>
<tr>
<th>Module</th>
<th>1</th>
<th>Surface Modeling Overview</th>
</tr>
</thead>
<tbody>
<tr>
<td>Module</td>
<td>2</td>
<td>Advanced Selection</td>
</tr>
<tr>
<td>Module</td>
<td>3</td>
<td>Advanced Datum Features</td>
</tr>
<tr>
<td>Module</td>
<td>4</td>
<td>Advanced Sketching</td>
</tr>
<tr>
<td>Module</td>
<td>5</td>
<td>Basic Surfacing Tools</td>
</tr>
<tr>
<td>Module</td>
<td>6</td>
<td>Boundary Blend Surfaces</td>
</tr>
</tbody>
</table>

### Day 2

<table>
<thead>
<tr>
<th>Module</th>
<th>7</th>
<th>Sweep Surfaces with Variable Sections</th>
</tr>
</thead>
<tbody>
<tr>
<td>Module</td>
<td>8</td>
<td>Helical Sweeps</td>
</tr>
<tr>
<td>Module</td>
<td>9</td>
<td>Swept Blends</td>
</tr>
<tr>
<td>Module</td>
<td>10</td>
<td>Analyzing Surface Curvature</td>
</tr>
<tr>
<td>Module</td>
<td>11</td>
<td>Additional Surface Analysis Tools</td>
</tr>
</tbody>
</table>

### Day 3

<table>
<thead>
<tr>
<th>Module</th>
<th>12</th>
<th>Extending and Trimming Surfaces</th>
</tr>
</thead>
<tbody>
<tr>
<td>Module</td>
<td>13</td>
<td>Manipulating Surfaces</td>
</tr>
<tr>
<td>Module</td>
<td>14</td>
<td>Creating and Editing Solids using Quilts</td>
</tr>
<tr>
<td>Module</td>
<td>15</td>
<td>Master Model Technique</td>
</tr>
<tr>
<td>Module</td>
<td>16</td>
<td>Project</td>
</tr>
</tbody>
</table>
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

Knowledge Check Questions

Module 3. Advanced Datum Features
   i. Creating Datum Graphs
   ii. Creating Datum Coordinate Systems
   iii. Creating Points On or Offset from Entities
   iv. Creating Points at Intersections
   v. Creating Points using an Offset Coordinate System
   vi. Sketching Geometry Datums
   vii. Creating Curves Through a Point or Vertex
   viii. Creating a Curve Through a Point Array
   ix. Creating a Curve from a Cross-Section
   x. Creating a Curve From Equation
   xi. Creating Composite Curves
   xii. Creating a Curve from Curve Intersections
   xiii. Creating a Curve at Surface Intersection
   xiv. Projecting and Wrapping Curves
   xv. Trimming Curves
   xvi. Creating Offset Curves
   xvii. Creating Cosmetic Sketches

Knowledge Check Questions

Module 4. Advanced Sketching
   i. Using Sketched Curves
   ii. Sketching Ellipses
   iii. Sketching Elliptical Fillets
   iv. Sketching Splines
   v. Modifying Splines — Basic Operations
   vi. Modifying Splines — Advanced Operations
   vii. Importing and Exporting Spline Points
viii. Sketching Conics
ix. Sketching Text
x. Analyzing Sketcher Convert Options
xi. Locking Sketcher Entities
xii. Analyzing Sketcher Dimension Options
xiii. 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

Knowledge Check Questions

Module 11. Additional Surface Analysis Tools
i. Using the Point Analysis Option
ii. Using the Radius Analysis Option
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

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

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

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 Design using Creo Parametric 2.0

Overview

Sheetmetal Design using Creo Parametric 2.0 is a comprehensive training course that teaches you how to create sheetmetal parts in Creo Parametric. The course builds upon the basic lessons you learned in Introduction to Creo Parametric 2.0 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 2.0

Audience

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

## Day 1

<table>
<thead>
<tr>
<th>Module</th>
<th>1</th>
<th>Introduction to the Creo Parametric Sheetmetal Design Process</th>
</tr>
</thead>
<tbody>
<tr>
<td>Module</td>
<td>2</td>
<td>Sheetmetal Model Fundamentals</td>
</tr>
<tr>
<td>Module</td>
<td>3</td>
<td>Creating Primary Sheetmetal Wall Features</td>
</tr>
<tr>
<td>Module</td>
<td>4</td>
<td>Creating Secondary Sheetmetal Wall Features</td>
</tr>
</tbody>
</table>

## Day 2

<table>
<thead>
<tr>
<th>Module</th>
<th>5</th>
<th>Bending and Unbending Sheetmetal Models</th>
</tr>
</thead>
<tbody>
<tr>
<td>Module</td>
<td>6</td>
<td>Modifying Sheetmetal Models</td>
</tr>
<tr>
<td>Module</td>
<td>7</td>
<td>Sheetmetal Setup and Tools</td>
</tr>
<tr>
<td>Module</td>
<td>8</td>
<td>Detailing Sheetmetal Designs</td>
</tr>
<tr>
<td>Module</td>
<td>9</td>
<td>Design Project</td>
</tr>
</tbody>
</table>
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. Creating Planar Bends
   iv. Creating Unbend Features
   v. Creating Bend Back Features
vi. Previewing and Creating Flat Patterns
vii. Creating Flat States
viii. Creating Split Area Features

Knowledge Check Questions

Module 6. Modifying Sheetmetal Models
i. Sheetmetal Cuts
ii. Notches and Punches
iii. Creating Corner Relief
iv. Die Form Features
v. Punch Form Features
vi. Utilizing Punch Model Annotations
vii. Creating Sketched Forms
viii. Flattening Forms and Unstamping Edges
ix. Creating Rip Features
x. Creating Edge Bends
xi. Joining Walls
xii. Patterning Walls
xiii. Mirroring Walls

Knowledge Check Questions

Module 7. 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 8. 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 9. Design Project
i. Designing a Stapler
Milling using Creo Parametric 2.0

Overview

Course Code TRN-3908-T
Course Length 5 Days

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

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

Course Objectives

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

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

Audience

• This course is intended for manufacturing engineers and NC machinists
# Agenda

## Day 1

<table>
<thead>
<tr>
<th>Module</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Introduction to Manufacturing</td>
</tr>
<tr>
<td>2</td>
<td>Creating Manufacturing Models</td>
</tr>
<tr>
<td>3</td>
<td>Configuring Operations</td>
</tr>
<tr>
<td>4</td>
<td>Using Reference Models</td>
</tr>
<tr>
<td>5</td>
<td>Using Workpiece Models</td>
</tr>
<tr>
<td>6</td>
<td>Creating and Using NC Model Assemblies</td>
</tr>
<tr>
<td>7</td>
<td>Creating and Configuring Workcells</td>
</tr>
</tbody>
</table>

## Day 2

<table>
<thead>
<tr>
<th>Module</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>8</td>
<td>Creating and Configuring Tools</td>
</tr>
<tr>
<td>9</td>
<td>Using Template Manufacturing Models</td>
</tr>
<tr>
<td>10</td>
<td>Using Manufacturing Parameters</td>
</tr>
<tr>
<td>11</td>
<td>Creating Face Milling Sequences</td>
</tr>
</tbody>
</table>

## Day 3

<table>
<thead>
<tr>
<th>Module</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>12</td>
<td>Creating Volume Milling Sequences</td>
</tr>
<tr>
<td>13</td>
<td>Creating Profile Milling Sequences</td>
</tr>
<tr>
<td>14</td>
<td>Creating Straight Cut Surface Milling Sequences</td>
</tr>
<tr>
<td>15</td>
<td>Creating From Surface Isolines Surface Milling Sequences</td>
</tr>
</tbody>
</table>

## Day 4

<table>
<thead>
<tr>
<th>Module</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>16</td>
<td>Creating Cut Line Surface Milling Sequences</td>
</tr>
<tr>
<td>17</td>
<td>Advanced Surface Milling Options</td>
</tr>
<tr>
<td>18</td>
<td>Creating Roughing and Re-roughing Sequences</td>
</tr>
<tr>
<td>19</td>
<td>Creating Finishing Sequences</td>
</tr>
</tbody>
</table>

## Day 5

<table>
<thead>
<tr>
<th>Module</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>20</td>
<td>Creating Trajectory Milling Sequences</td>
</tr>
<tr>
<td>21</td>
<td>Creating Holemaking Sequences</td>
</tr>
<tr>
<td>Module</td>
<td>22</td>
</tr>
<tr>
<td>--------</td>
<td>----</td>
</tr>
<tr>
<td>Module</td>
<td>23</td>
</tr>
<tr>
<td>Module</td>
<td>24</td>
</tr>
</tbody>
</table>
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 Workcells
   i. Creating and Configuring Workcells
   Knowledge Check Questions

Module 8. Creating and Configuring Tools
   i. Understanding Milling Tools
   ii. Creating Standard Milling Tools
   iii. Creating Solid Model Milling Tools
   iv. Creating and Using Tool Cutting Data
   v. Retrieving Tool Data
   Knowledge Check Questions

Module 9. Using Template Manufacturing Models
   i. Using Template Manufacturing Models
   Knowledge Check Questions

Module 10. Using Manufacturing Parameters
   i. Understanding Manufacturing Parameter Concepts
   ii. Configuring Parameter Values
   iii. Using Site Parameter Files
   Knowledge Check Questions
Module 11. Creating Face Milling Sequences
   i. Basic Face Milling
   ii. Lateral Control Face Milling Parameters
   iii. Depth Control Face Milling Parameters
   iv. Entry and Exit Face Milling Parameters

Knowledge Check Questions

Module 12. Creating Volume Milling Sequences
   i. Basic Volume Milling
   ii. Volume Milling with Mill Windows
   iii. Scanning Volume Milling Parameters
   iv. Depth and Lateral Control Volume Milling Parameters
   v. Stock Allowance Volume Milling Parameters
   vi. Gathering Mill Volumes
   vii. Modifying Volume Milling Toolpaths

Knowledge Check Questions

Module 13. Creating Profile Milling Sequences
   i. Basic Profile Milling
   ii. Depth and Lateral Control Profile Milling Parameters
   iii. Lead In and Lead Out Motions

Knowledge Check Questions

Module 14. Creating Straight Cut Surface Milling Sequences
   i. Understanding Surface Milling
   ii. Straight Cut Surface Milling
   iii. Straight Cut Surface Milling Parameters
   iv. Creating Surface Milling Reference Geometry

Knowledge Check Questions

Module 15. Creating From Surface Isolines Surface Milling Sequences
   i. From Surface Isolines Surface Milling

Knowledge Check Questions

Module 16. Creating Cut Line Surface Milling Sequences
   i. Cut Line Surface Milling

Knowledge Check Questions

Module 17. Advanced Surface Milling Options
   i. Advanced Surface Milling Options

Knowledge Check Questions

Module 18. Creating Roughing and Re-roughing Sequences
   i. Basic Roughing and Re-roughing
   ii. Roughing Scans and Entry and Exit Parameters
iii. Step Depth and Tolerance Control Roughing Parameters
iv. Additional Scallop Height Control Roughing Parameters
v. Roughing Corner Options

Knowledge Check Questions

Module 19. Creating Finishing Sequences
i. Basic Finishing
ii. Editing Finishing Parameters

Knowledge Check Questions

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

Knowledge Check Questions

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

Knowledge Check Questions

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

Knowledge Check Questions

Module 23. Using the Process Manager
i. Using Process Manager Tools
ii. Editing Process Items
iii. Creating New Items in the Process Manager
iv. Creating and Using Manufacturing Templates

Knowledge Check Questions

Module 24. Creating and Post-Processing CL Data Files
i. Creating and Post-Processing CL Data Files

Knowledge Check Questions
Cabling using Creo Parametric 2.0

Overview

Course Code: TRN-3909-T
Course Length: 3 Days

In this course, you will learn how to create 3-D electrical harnesses using Creo Parametric 2.0. This includes using Creo Schematics to pass schematic diagram information into the 3-D harness designs created within Creo Parametric 2.0. 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 2.0. 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 2.0.

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

Course Objectives

- Learn the basic Creo Parametric 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 flat harness
• Document harness designs
• Comprehensive design project

**Prerequisites**

• Introduction to Creo Parametric 2.0 or equivalent experience
• Introduction to Creo Schematics 2.0 (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

<table>
<thead>
<tr>
<th>Module</th>
<th>1</th>
<th>Introduction to the Creo Basic Cabling Process</th>
</tr>
</thead>
<tbody>
<tr>
<td>Module</td>
<td>2</td>
<td>Creating Harness Assembly Structures</td>
</tr>
<tr>
<td>Module</td>
<td>3</td>
<td>Setting Up for Cabling</td>
</tr>
<tr>
<td>Module</td>
<td>4</td>
<td>Routing Wires and Cables</td>
</tr>
<tr>
<td>Module</td>
<td>5</td>
<td>Modifying Wire Routings</td>
</tr>
</tbody>
</table>

## Day 2

<table>
<thead>
<tr>
<th>Module</th>
<th>6</th>
<th>Routing and Utilizing Networks</th>
</tr>
</thead>
<tbody>
<tr>
<td>Module</td>
<td>7</td>
<td>Establishing Logical References</td>
</tr>
<tr>
<td>Module</td>
<td>8</td>
<td>Routing Wires and Cables using Logical Data</td>
</tr>
<tr>
<td>Module</td>
<td>9</td>
<td>Creating Harness Components and Cosmetics</td>
</tr>
<tr>
<td>Module</td>
<td>10</td>
<td>Creating Flat Harnesses</td>
</tr>
</tbody>
</table>

## Day 3

<table>
<thead>
<tr>
<th>Module</th>
<th>11</th>
<th>Documenting Harness Designs</th>
</tr>
</thead>
<tbody>
<tr>
<td>Module</td>
<td>12</td>
<td>Project (Creo Schematics-Based)</td>
</tr>
<tr>
<td>Module</td>
<td>13</td>
<td>Project (Manual Routing)</td>
</tr>
</tbody>
</table>
Course Content

Module 1. Introduction to the Creo 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

Knowledge Check Questions

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

Knowledge Check Questions

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. Creating Wire Spools
  vi. Creating Cable Spools
  vii. Creating Ribbon Cable Spools

Knowledge Check Questions

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

Knowledge Check Questions

Module 5. Modifying Wire Routings
  i. Deleting Wires and Segments
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

Knowledge Check Questions

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

Knowledge Check Questions

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

Knowledge Check Questions

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

Knowledge Check Questions

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

Knowledge Check Questions

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

Knowledge Check Questions

Module 12. Project (Creo Schematics-Based)  
i. The Electrical Cabinet  
ii. Set Up for Cabling  
iii. Routing Wires and Cables  
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  
v. Flattening the Harness  
vi. Creating a Harness Drawing
Piping using Creo Parametric 2.0

Overview

Course Code: TRN-3910-T
Course Length: 3 Days

In this course, you will learn how to manually create (non-specification driven) mechanical piping designs using Creo Parametric 2.0. This includes learning how to configure pipelines, how to route pipelines, and how to insert pipe fittings such as valves and reducers. You will also learn how to create specification driven industrial piping designs using Creo Parametric 2.0. This includes learning how to use schematic diagrams created with Creo Schematics to drive 3-D industrial piping designs created within Creo Parametric 2.0. Finally, you learn how to document piping designs by creating drawings that include BOM tables, pipe bend tables, and engineering information, as well as how to export ISOGEN format files for creating pipeline, spool and systems isometric drawings.

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

Course Objectives

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

- Introduction to Creo Parametric 2.0 or equivalent experience

Audience

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

## Day 1

<table>
<thead>
<tr>
<th>Module</th>
<th>1</th>
<th>Introduction to Piping</th>
</tr>
</thead>
<tbody>
<tr>
<td>Module</td>
<td>2</td>
<td>Creating Piping Assembly Structures</td>
</tr>
<tr>
<td>Module</td>
<td>3</td>
<td>Configuring and Routing Pipelines</td>
</tr>
<tr>
<td>Module</td>
<td>4</td>
<td>Moving and Modifying Pipelines</td>
</tr>
<tr>
<td>Module</td>
<td>5</td>
<td>Configuring and Inserting Fittings</td>
</tr>
</tbody>
</table>

## Day 2

<table>
<thead>
<tr>
<th>Module</th>
<th>6</th>
<th>Creating Solid Pipeline Models</th>
</tr>
</thead>
<tbody>
<tr>
<td>Module</td>
<td>7</td>
<td>Gathering Piping Information</td>
</tr>
<tr>
<td>Module</td>
<td>8</td>
<td>Creating Piping Drawings</td>
</tr>
<tr>
<td>Module</td>
<td>9</td>
<td>Specification Database Overview</td>
</tr>
<tr>
<td>Module</td>
<td>10</td>
<td>Setting Up Specification Databases: Piping</td>
</tr>
<tr>
<td>Module</td>
<td>11</td>
<td>Setting Up Specification Databases: Fittings</td>
</tr>
</tbody>
</table>

## Day 3

<table>
<thead>
<tr>
<th>Module</th>
<th>12</th>
<th>General Master Catalog Files</th>
</tr>
</thead>
<tbody>
<tr>
<td>Module</td>
<td>13</td>
<td>Configuring Project-Specific Data Files</td>
</tr>
<tr>
<td>Module</td>
<td>14</td>
<td>Specification-Driven Routing and Inserting Fittings</td>
</tr>
<tr>
<td>Module</td>
<td>15</td>
<td>Using Creo Schematics Process and Instrumentation Diagrams Data</td>
</tr>
<tr>
<td>Module</td>
<td>16</td>
<td>Schematic Driven Pipeline Modeling</td>
</tr>
<tr>
<td>Module</td>
<td>17</td>
<td>Using ISOGEN PCF Data</td>
</tr>
</tbody>
</table>
Course Content

**Module 1. Introduction to Piping**
- i. Understanding Piping Design Methods
- ii. Manual Piping Development Process
- iii. Specification-Driven Piping Development Process
- iv. Understanding Piping Terminology
- v. Understanding 2-D Schematic Piping Designs

**Knowledge Check Questions**

**Module 2. Creating Piping Assembly Structures**
- i. Planning Piping Assembly Structures
- ii. Piping Assembly Structure: Sub-Assembly
- iii. Piping Assembly Structure: No Sub-Assembly
- iv. Piping Assembly Structure: Sub-Assemblies at Top Level
- v. Understanding Piping Large Assembly Management Tools
- vi. Creating and Configuring Piping Skeletons
- vii. Sharing Routing Geometry
- viii. Assembling Piping Components Using Component Interfaces

**Knowledge Check Questions**

**Module 3. Configuring and Routing Pipelines**
- i. Understanding Pipeline Routing
- ii. Configuring Non-Specification-Driven Pipelines
- iii. Routing Pipelines
- iv. Routing Flexible Hoses

**Knowledge Check Questions**

**Module 4. Moving and Modifying Pipelines**
- i. Adding Points and Moving Pipe Segments
- ii. Modifying Pipeline Routing

**Knowledge Check Questions**

**Module 5. Configuring and Inserting Fittings**
- i. Understanding Fittings
- ii. Creating Fittings
- iii. Inserting Fittings

**Knowledge Check Questions**

**Module 6. Creating Solid Pipeline Models**
- i. Creating Solid Pipes
- ii. Extracting Models

**Knowledge Check Questions**

**Module 7. Gathering Piping Information**
Module 8. Creating Piping Drawings
i. Creating Piping Drawings
ii. Displaying Piping Report Parameters

Module 9. Specification Database Overview
i. Understanding the Piping Specification Database
ii. Master Catalog Directory and File Structure: Pipes
iii. Master Catalog Directory and File Structure: Fittings

Module 10. Setting Up Specification Databases: Piping
i. Piping Master Catalog Directory File
ii. Pipe Design Master Catalog Files
iii. Pipe Manufacture Directory File
iv. Pipe Manufacture Master Catalog Files
v. Bend Machine Master Catalog Files

i. Fitting Master Catalog Files
ii. Mapping Catalog Data to Fittings
iii. Understanding Fitting Libraries and Fitting Models
iv. Configuring Specification-Driven Fittings
v. Bolt Nut Master Catalog Files

Module 12. General Master Catalog Files
i. Master Catalog Files: Piping Material File
ii. End Compatibility File
iii. Insulation Files
iv. Combined Size Code File
v. Units System File

Module 13. Configuring Project-Specific Data Files
i. Configuring Auto-Selection Files
ii. Configuring Specification Directory Files
iii. Insulation Directory Files
iv. Fitting Category Map Files
v. Configuring Piping Config.pro Options

Knowledge Check Questions
Module 14. Specification-Driven Routing and Inserting Fittings
   i. Specification-Driven Pipeline Routing
   ii. Specification-Driven Fitting Insertion
   iii. Design Rules
   iv. Routing Continuous Fittings
   v. Non-Break Fittings

Knowledge Check Questions

Module 15. Using Creo Schematics Process and Instrumentation Diagrams Data
   i. Process and Instrumentation Diagrams Overview
   ii. Creo Schematics Concepts and Terminology
   iii. Investigating Process and Instrumentation Diagrams
   iv. Exporting Design Information

Knowledge Check Questions

Module 16. Schematic Driven Pipeline Modeling
   i. Designating Schematic Information
   ii. Routing Pipelines Using Schematic Information
   iii. Inserting Fittings Using Schematic Information
   iv. Schematic Consistency Check Report
   v. Updating Piping Designs from Creo Schematics

Knowledge Check Questions

Module 17. Using ISOGEN PCF Data
   i. Creating ISOGEN PCF Data

Knowledge Check Questions
Introduction to Creo Simulate 2.0

Overview

<table>
<thead>
<tr>
<th>Course Code</th>
<th>TRN-3911-0</th>
</tr>
</thead>
<tbody>
<tr>
<td>Course Length</td>
<td>5 days</td>
</tr>
</tbody>
</table>

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

- Learn the basic Simulate analysis process
- Learn theory and simulate model topics
- Explore results
- Learn about 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
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

<table>
<thead>
<tr>
<th>Module</th>
<th>1</th>
<th>Introduction to Creo Simulate</th>
</tr>
</thead>
<tbody>
<tr>
<td>Module</td>
<td>2</td>
<td>Theoretical Foundations</td>
</tr>
<tr>
<td>Module</td>
<td>3</td>
<td>Simulation Models</td>
</tr>
<tr>
<td>Module</td>
<td>4</td>
<td>Materials and Material Properties</td>
</tr>
<tr>
<td>Module</td>
<td>5</td>
<td>Structural Constraints</td>
</tr>
<tr>
<td>Module</td>
<td>6</td>
<td>Structural Loads</td>
</tr>
</tbody>
</table>

## Day 2

<table>
<thead>
<tr>
<th>Module</th>
<th>7</th>
<th>Meshing</th>
</tr>
</thead>
<tbody>
<tr>
<td>Module</td>
<td>8</td>
<td>Convergence</td>
</tr>
<tr>
<td>Module</td>
<td>9</td>
<td>Structural Analysis</td>
</tr>
<tr>
<td>Module</td>
<td>10</td>
<td>Introduction to Results Evaluation</td>
</tr>
</tbody>
</table>

## Day 3

<table>
<thead>
<tr>
<th>Module</th>
<th>11</th>
<th>Refining the Design</th>
</tr>
</thead>
<tbody>
<tr>
<td>Module</td>
<td>12</td>
<td>Basic Model Debugging</td>
</tr>
<tr>
<td>Module</td>
<td>13</td>
<td>Singularities</td>
</tr>
</tbody>
</table>

## Day 4

<table>
<thead>
<tr>
<th>Module</th>
<th>14</th>
<th>Analyzing Assemblies</th>
</tr>
</thead>
<tbody>
<tr>
<td>Module</td>
<td>15</td>
<td>Shells</td>
</tr>
<tr>
<td>Module</td>
<td>16</td>
<td>Idealizations</td>
</tr>
</tbody>
</table>

## Day 5

<table>
<thead>
<tr>
<th>Module</th>
<th>17</th>
<th>Thermal Analysis</th>
</tr>
</thead>
<tbody>
<tr>
<td>Module</td>
<td>18</td>
<td>Advanced Analysis</td>
</tr>
<tr>
<td>Module</td>
<td>19</td>
<td>Project</td>
</tr>
</tbody>
</table>
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. 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
Introduction to Creo Schematics 2.0

Overview

Course Code: TRN-3912-T
Course Length: 3 Days

In this course, you will learn how to use Creo Schematics to create schematic diagrams for both electrical harness and piping designs. You will also learn how to administer the Creo Schematics working environment, configure the catalog library, and create functional block diagrams and block interconnect diagrams. In addition, you will learn how to create electrical circuit diagrams, wiring diagrams (including wire interconnect diagrams), and process and instrumentation design diagrams for piping systems. Finally, you will learn how to use wiring diagrams and process and instrumentation design diagrams to configure 3-D harness designs and industrial piping designs created within 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

• Understand the 2-D schematic design processes for electrical harness designs and piping designs
• Configure the Creo Schematics working environment
• Create and configure the catalog library
• Create functional block diagrams (including block interconnect diagrams)
• Create circuit diagrams
• Create wiring diagrams (including wire interconnect diagrams)
• Create process and instrumentation design diagrams
• Communicate diagram information to Creo Parametric
Prerequisites

- None

Audience

- This course is intended for engineers involved in the schematic 2-D layout of either electrical diagrams or piping diagrams
# Agenda

## Day 1

<table>
<thead>
<tr>
<th>Module</th>
<th>1</th>
<th>Introduction to Creo Schematics</th>
</tr>
</thead>
<tbody>
<tr>
<td>Module</td>
<td>2</td>
<td>Configuring the Working Environment</td>
</tr>
<tr>
<td>Module</td>
<td>3</td>
<td>Configuring Designs and Diagram Sheets</td>
</tr>
<tr>
<td>Module</td>
<td>4</td>
<td>Configuring Catalog Properties</td>
</tr>
<tr>
<td>Module</td>
<td>5</td>
<td>Creating Design Templates and Template Sheets</td>
</tr>
<tr>
<td>Module</td>
<td>6</td>
<td>Using Creo Schematics Tools</td>
</tr>
</tbody>
</table>

## Day 2

<table>
<thead>
<tr>
<th>Module</th>
<th>7</th>
<th>Creating Catalog Artifacts</th>
</tr>
</thead>
<tbody>
<tr>
<td>Module</td>
<td>8</td>
<td>Creating and Configuring a Central Catalog</td>
</tr>
<tr>
<td>Module</td>
<td>9</td>
<td>Reviewing Designs</td>
</tr>
<tr>
<td>Module</td>
<td>10</td>
<td>Instancing and Manipulating Objects</td>
</tr>
<tr>
<td>Module</td>
<td>11</td>
<td>Creating Block Diagrams</td>
</tr>
</tbody>
</table>

## Day 3

<table>
<thead>
<tr>
<th>Module</th>
<th>12</th>
<th>Creating Circuit Diagrams</th>
</tr>
</thead>
<tbody>
<tr>
<td>Module</td>
<td>13</td>
<td>Creating Wiring Diagrams</td>
</tr>
<tr>
<td>Module</td>
<td>14</td>
<td>Creating Interconnect Diagrams</td>
</tr>
<tr>
<td>Module</td>
<td>15</td>
<td>Creating P&amp;ID Diagrams</td>
</tr>
<tr>
<td>Module</td>
<td>16</td>
<td>Using Schematic Design Data in Creo Parametric Harness and Piping Designs</td>
</tr>
<tr>
<td>Module</td>
<td>17</td>
<td>Using Pro/Diagram Information</td>
</tr>
</tbody>
</table>
Course Content

Module 1. Introduction to Creo Schematics
   i. Electrical Diagram and Harness Design Process
   ii. Piping Design Process
   iii. Understanding Diagram Types
   iv. Understanding Creo Schematics Concepts and Terminology

Knowledge Check Questions

Module 2. Configuring the Working Environment
   i. Understanding the Creo Schematics User Interface
   ii. Configuring Options and Switches
   iii. Configuring Colors

Knowledge Check Questions

Module 3. Configuring Designs and Diagram Sheets
   i. Creating and Configuring Designs and Diagram Sheets
   ii. Configuring Grids
   iii. Configuring Layers
   iv. Configuring Location Sets

Knowledge Check Questions

Module 4. Configuring Catalog Properties
   i. Configuring Object Types and Global Parameters
   ii. Configuring Derived Parameters

Knowledge Check Questions

Module 5. Creating Design Templates and Template Sheets
   i. Creating and Configuring Template Sheets
   ii. Creating and Using Design Templates

Knowledge Check Questions

Module 6. Using Creo Schematics Tools
   i. Using Selection Tools
   ii. Using Viewing Tools
   iii. Understanding ProbeSpecifier Tools

Knowledge Check Questions

Module 7. Creating Catalog Artifacts
   i. Understanding Catalog Artifacts
   ii. Using Datatables and Datasets
   iii. Creating Ports
   iv. Creating Blocks
   v. Creating Fibers
   vi. Understanding Groups
vii. Creating Groups
viii. Creating Cables
ix. Exporting and Importing Design Items

Knowledge Check Questions

Module 8. Creating and Configuring a Central Catalog
   i. Creating a Central Catalog
   ii. Managing Central Catalogs

Knowledge Check Questions

Module 9. Reviewing Designs
   i. Using the Model Explorer

Knowledge Check Questions

Module 10. Instancing and Manipulating Objects
   i. Instancing Objects
   ii. Manipulating Objects

Knowledge Check Questions

Module 11. Creating Block Diagrams
   i. Creating Block Diagrams

Knowledge Check Questions

Module 12. Creating Circuit Diagrams
   i. Creating Circuit Diagrams
   ii. Managing Signal Networks

Knowledge Check Questions

Module 13. Creating Wiring Diagrams
   i. Creating Wiring Diagrams

Knowledge Check Questions

Module 14. Creating Interconnect Diagrams
   i. Understanding Interconnect Diagrams
   ii. Creating Block Interconnect Diagrams
   iii. Creating Wiring Interconnect Diagrams
   iv. Verifying Interconnect Diagrams

Knowledge Check Questions

Module 15. Creating P&ID Diagrams
   i. Creating P&ID Diagrams
   ii. Updating P&ID Diagrams

Knowledge Check Questions

   i. Transferring XML Data into 3-D Harness Designs
   ii. Understanding 3-D Piping Design Approaches
iii. Transferring XML Data into 3-D Piping Designs
iv. Verifying Creo Schematics Design Data

Knowledge Check Questions

Module 17. Using Pro/Diagram Information
i. Migrating Pro/DIAGRAM Schematics

Knowledge Check Questions
Introduction to Creo Layout 2.0

Overview

Course Code: TRN-3913-T
Course Length: 1 Day

In this course, you will learn about a new Creo module called Layout 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 makes use of precision panels and sketching guides to intelligently sketch a variety of 2-D geometry, as well as learn how to manipulate, organize, and import layout geometry. Finally, you will learn how to leverage 2-D Layout designs in 3-D models.

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

Course Objectives

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

- Introduction to Creo Parametric 2.0 (optional)

**Audience**

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

### Day 1

<table>
<thead>
<tr>
<th>Module</th>
<th>Number</th>
<th>Topic</th>
</tr>
</thead>
<tbody>
<tr>
<td>Module 1</td>
<td>1</td>
<td>Introduction to Layout</td>
</tr>
<tr>
<td>Module 2</td>
<td>2</td>
<td>Layout Sketching Methodology</td>
</tr>
<tr>
<td>Module 3</td>
<td>3</td>
<td>Sketching Layout Geometry</td>
</tr>
<tr>
<td>Module 4</td>
<td>4</td>
<td>Manipulating Layout Geometry</td>
</tr>
<tr>
<td>Module 5</td>
<td>5</td>
<td>Organizing Layout Geometry</td>
</tr>
<tr>
<td>Module 6</td>
<td>6</td>
<td>Importing Layout Geometry</td>
</tr>
<tr>
<td>Module 7</td>
<td>7</td>
<td>Annotating Layouts</td>
</tr>
<tr>
<td>Module 8</td>
<td>8</td>
<td>Utilizing Layouts in 3-D Models</td>
</tr>
</tbody>
</table>
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 Precision Panels
   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 Conic Arcs
   xi. Sketching Splines
   xii. Sketching Centerlines and Center Points
   xiii. 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 Layout Geometry
   iv. Translating, Rotating, and Scaling Layout Geometry
   v. Importing 2-D Geometry from the Palette
   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

*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

*Knowledge Check Questions*

**Module 8. Utilizing Layouts in 3-D Models**
- i. Designating Public Geometry
- ii. Assembling Layouts
- iii. Utilizing Layouts in 3-D Models

*Knowledge Check Questions*
Introduction to Creo Illustrate 2.0

Overview

Course Code: TRN-3914-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

<table>
<thead>
<tr>
<th>Module</th>
<th>1</th>
<th>Introduction to Creo Illustrate</th>
</tr>
</thead>
<tbody>
<tr>
<td>Module</td>
<td>2</td>
<td>Create an Illustration and Work with Figures</td>
</tr>
<tr>
<td>Module</td>
<td>3</td>
<td>Exploding Assemblies</td>
</tr>
<tr>
<td>Module</td>
<td>4</td>
<td>Authoring Sequenced Process Steps</td>
</tr>
<tr>
<td>Module</td>
<td>5</td>
<td>Creo Illustrate Animations</td>
</tr>
<tr>
<td>Module</td>
<td>6</td>
<td>Create and Manage Annotations, Sub-Assemblies, and Parts Lists</td>
</tr>
<tr>
<td>Module</td>
<td>7</td>
<td>Publishing and Exporting 3-D Illustrations</td>
</tr>
</tbody>
</table>
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 File Menu
  vii. The Creo Illustrate Quick Access Toolbar
  viii. The Creo Illustrate Primary Panel
  ix. The Creo Illustrate Upper Data Panel
  x. The Creo Illustrate Lower Data Panel
  xi. Creo Illustrate Structure Tree
  xii. The Creo Illustrate Status Bar
  xiii. Starting Creo Illustrate
  xiv. Working with Creo Illustrate

Knowledge Check Questions

Module 2. Create an Illustration and Work with Figures
  i. Import 3-D MCAD Data
  ii. Display Parts in the Structure Edit Viewing Area
  iii. Structure Edit Mode
  iv. Creo Illustrate Ribbon Home Tab
  v. Advanced sBOM Editing Tools
  vi. Working with Figures
  vii. sBOM Structure versus Displayed Parts and Sub-Assemblies
  viii. Figure Orientation
  ix. Selecting Parts and Sub-Assemblies
  x. Find Parts and Sub-Assemblies
  xi. Changing the Figure Display
  xii. Figure Rendering Options
  xiii. Hide and Unhide Parts and Sub-Assemblies
  xiv. Filters

Knowledge Check Questions

Module 3. Exploding Assemblies
  i. Explode Figures
  ii. Translate Exploded Figures
  iii. Rotate 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. 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
iii. E-mailing a Figure

Knowledge Check Questions
Flexible Modeling using Creo Parametric 2.0

Overview

<table>
<thead>
<tr>
<th>Course Code</th>
<th>TRN-3915-T</th>
</tr>
</thead>
<tbody>
<tr>
<td>Course Length</td>
<td>1 Day</td>
</tr>
</tbody>
</table>

In this course, you will learn how to use flexible modeling tools to edit existing geometry on parametric models. The flexible modeling process typically involves initially selecting model surfaces, then refining the selected surface set using smart selection tools, and finally modifying the selected geometry by applying transformation tools, patterning tools, or symmetry tools. At the end of each module, you will complete a set of review questions to reinforce critical topics from that module. At the end of the course, you will complete a course assessment in Pro/FICIENCY intended to evaluate your understanding of the course as a whole.

Course Objectives

- Understand flexible modeling basics
- Apply selection and tools
- Utilize editing and transformations
- Work with recognition
- Use propagation and other editing features

Prerequisites

- Introduction to Creo Parametric 2.0 or equivalent experience

Audience

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

### Day 1

<table>
<thead>
<tr>
<th>Module</th>
<th>1</th>
<th>Introduction to Flexible Modeling</th>
</tr>
</thead>
<tbody>
<tr>
<td>Module</td>
<td>2</td>
<td>Selection and Tools</td>
</tr>
<tr>
<td>Module</td>
<td>3</td>
<td>Editing and Transformations</td>
</tr>
<tr>
<td>Module</td>
<td>4</td>
<td>Recognition</td>
</tr>
<tr>
<td>Module</td>
<td>5</td>
<td>Propagation and Other Editing Features</td>
</tr>
</tbody>
</table>
Course Content

Module 1. Introduction to Flexible Modeling
   i. Understanding Flexible Modeling
   ii. Understanding the Flexible Modeling User Interface
   iii. The Flexible Modeling Process

Knowledge Check Questions

Module 2. Selection and Tools
   i. Using the Selection Filter
   ii. Applying Shape Selection
   iii. Applying Boss Selections
   iv. Applying Cut Selections
   v. Applying Round Selections
   vi. Leveraging Geometry Rules
   vii. Right-Clicking for Quick Access to Tools

Knowledge Check Questions

Module 3. Editing and Transformations
   i. Applying Flexible Move using the Dragger
   ii. Applying Flexible Move by Dimension
   iii. Applying Flexible Move using Constraints
   iv. Applying Flexible Offset
   v. Modifying Analytic Geometry
   vi. Using Flexible Mirror
   vii. Applying Flexible Substitute
   viii. Using the Edit Round Feature

Knowledge Check Questions

Module 4. Recognition
   i. Working with Pattern Recognition
   ii. Working with Symmetry Recognition

Knowledge Check Questions

Module 5. Propagation and Other Editing Features
   i. Using the Flexible Attach Feature
   ii. Applying the Flexible Remove Feature
   iii. Combining Transform with Recognition Features

Knowledge Check Questions
Introduction to Creo Direct 2.0

Overview

Course Code  TRN-3916-T
Course Length  1 Day

In this course, you will learn direct modeling using Creo Direct 2.0. 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 by extruding, revolving, and sweeping sketches and sketch regions
- Create datums, holes, rounds, chamfers, and draft
- Use advanced sketching techniques
- Rapidly select and directly modify 3-D shapes in your models
- Use advanced tools including sweeps and shells
- Understand the direct approach to creating assemblies
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

<table>
<thead>
<tr>
<th>Module</th>
<th>1</th>
<th>Introduction to Creo Direct</th>
</tr>
</thead>
<tbody>
<tr>
<td>Module</td>
<td>2</td>
<td>Creating Sketches in 2-D Mode</td>
</tr>
<tr>
<td>Module</td>
<td>3</td>
<td>Extruding and Revolving Sketches and Sketch Regions</td>
</tr>
<tr>
<td>Module</td>
<td>4</td>
<td>Datums, Holes, Rounds, Chamfers, and Drafts</td>
</tr>
<tr>
<td>Module</td>
<td>5</td>
<td>Advanced Sketching</td>
</tr>
<tr>
<td>Module</td>
<td>6</td>
<td>Selecting, Modifying, and Reusing 3-D Geometry</td>
</tr>
<tr>
<td>Module</td>
<td>7</td>
<td>Advanced Tools: Sweep and Shell</td>
</tr>
<tr>
<td>Module</td>
<td>8</td>
<td>Creating Assemblies in Creo Direct</td>
</tr>
</tbody>
</table>
Course Content

Module 1. Introduction to Creo Direct
   i. Understanding Direct Modeling
   ii. Understanding the User Interface
   iii. Opening and Creating a Creo Direct Model
   iv. Orienting and Positioning the Model
   v. Understanding Selection

Knowledge Check Questions

Module 2. Creating Sketches in 2-D Mode
   i. Understanding 2-D Mode
   ii. Configuring the 2-D Grid
   iii. Understanding the Precision Panel
   iv. Sketching a Line/Arc Chain
   v. Sketching Rectangles, Circles, and Arcs
   vi. Using Guides

Knowledge Check Questions

Module 3. Extruding and Revolving Sketches and Sketch Regions
   i. Understanding Sketches and Sketch Regions
   ii. Creating Extrusions
   iii. Revolving Sketches
   iv. Understanding Snap to Geometry

Knowledge Check Questions

Module 4. Datums, Holes, Rounds, Chamfers, and Drafts
   i. Creating Datums
   ii. Creating Holes
   iii. Creating Rounds and Chamfers
   iv. Creating Drafts

Knowledge Check Questions

Module 5. Advanced Sketching
   i. Creating Construction Geometry
   ii. Trimming Sketched Entities
   iii. Projecting 3-D Geometry Into a Sketch
   iv. Mirroring Sketched Entities

Knowledge Check Questions

Module 6. Selecting, Modifying, and Reusing 3-D Geometry
   i. Using Shape Selection
   ii. Shape Selection Types
   iii. Using Marquee Selection
   iv. Leveraging Geometry Selection Rules
v. Understanding the CoPilot
vi. Using the Move/Rotate Tool
vii. Moving Geometry by Dimension
viii. Editing Rounds
ix. Modifying Analytic Surfaces
x. Removing Surfaces From a Solid
xi. Patterning Geometry

Knowledge Check Questions

Module 7. Advanced Tools: Sweep and Shell
i. Creating Sweeps
ii. Shelling Solid Geometry

Knowledge Check Questions

Module 8. Creating Assemblies in Creo Direct
i. Assembling Components
ii. Designing Components in Assembly

Knowledge Check Questions
Introduction to Creo Options Modeler 2.0

Overview

<table>
<thead>
<tr>
<th>Course Code</th>
<th>TRN-3917-T</th>
</tr>
</thead>
<tbody>
<tr>
<td>Course Length</td>
<td>1 Day</td>
</tr>
</tbody>
</table>

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

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

Course Objectives

- Understand and use Creo Options Modeler
- Create interchange assemblies
- Create configurable modules and products
- Define options and choices
- Utilize configurable assemblies
Prerequisites

- Introduction to Creo Parametric 2.0
- Advanced Assembly Design using 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>Introduction to Creo Options Modeler</th>
</tr>
</thead>
<tbody>
<tr>
<td>Module</td>
<td>2</td>
<td>Creating Interchange Assemblies</td>
</tr>
<tr>
<td>Module</td>
<td>3</td>
<td>Creating Configurable Modules and Products</td>
</tr>
<tr>
<td>Module</td>
<td>4</td>
<td>Defining Options and Choices</td>
</tr>
<tr>
<td>Module</td>
<td>5</td>
<td>Utilizing Configurable Assemblies</td>
</tr>
<tr>
<td>Module</td>
<td>6</td>
<td>Project</td>
</tr>
</tbody>
</table>
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. Project
   i. Designing Variations for the Power Head
   ii. The Drill Variation
   iii. The Pump Variation
   iv. The Auger Variation
Overview

The Mold application provides the tools to create a mold model from start to finish using the mold design process within Creo Parametric 2.0. In this course, you will learn how to create, modify, and analyze mold components and assemblies. Any changes made to the design model automatically propagate to the mold components and assemblies. You will also how to create final extract components that reflect the geometry of the design model, along with shrinkage considerations, adequate drafting, mold features, and cooling systems. After completing this course, you will have a better understanding of the mold design process and how to create molded products using the mold design 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 Pro/FICIENCY intended to evaluate your understanding of the course as a whole.

Course Objectives

• Learn the basic mold process
• Prepare design models for the mold process
• Analyze design models to ensure their readiness for molding
• Create mold models
• Apply shrinkage to the reference model
• Create and assemble workpieces into the mold model
• Create mold volumes
• Create parting lines and parting surfaces
• Split mold volumes
• Extract mold components
• Create mold features
• Learn how to fill and open the mold

Prerequisites

• Introduction to Creo Parametric 2.0
• Basic understanding of industry standard Mold design terminology and processes
• Knowledge of Creo Parametric surfacing techniques a plus
Audience

- This course is intended for designers, machinists, and manufacturing engineers. People in related roles will also benefit from taking this course.
**Agenda**

**Day 1**

<table>
<thead>
<tr>
<th>Module</th>
<th>1</th>
<th>Introduction to the Creo Parametric Basic Mold Process</th>
</tr>
</thead>
<tbody>
<tr>
<td>Module</td>
<td>2</td>
<td>Design Model Preparation</td>
</tr>
<tr>
<td>Module</td>
<td>3</td>
<td>Design Model Analysis</td>
</tr>
<tr>
<td>Module</td>
<td>4</td>
<td>Mold Models</td>
</tr>
<tr>
<td>Module</td>
<td>5</td>
<td>Shrinkage</td>
</tr>
<tr>
<td>Module</td>
<td>6</td>
<td>Workpieces</td>
</tr>
<tr>
<td>Module</td>
<td>7</td>
<td>Mold Volume Creation</td>
</tr>
</tbody>
</table>

**Day 2**

<table>
<thead>
<tr>
<th>Module</th>
<th>8</th>
<th>Parting Line and Parting Surface Creation</th>
</tr>
</thead>
<tbody>
<tr>
<td>Module</td>
<td>9</td>
<td>Splitting Mold Volumes</td>
</tr>
<tr>
<td>Module</td>
<td>10</td>
<td>Mold Component Extraction</td>
</tr>
<tr>
<td>Module</td>
<td>11</td>
<td>Mold Features Creation</td>
</tr>
<tr>
<td>Module</td>
<td>12</td>
<td>Filling and Opening the Mold</td>
</tr>
</tbody>
</table>
Course Content

Module 1. Introduction to the Creo Parametric Basic Mold Process
   i. Creo Parametric Basic Mold Process

Knowledge Check Questions

Module 2. Design Model Preparation
   i. Understanding Mold Theory
   ii. Preparing Design Models for the Mold Process
   iii. Creating Profile Rib Features
   iv. Creating Drafts Split at Sketch
   v. Creating Drafts Split at Curve
   vi. Creating Drafts Split at Surface

Knowledge Check Questions

Module 3. Design Model Analysis
   i. Analyzing Design Models Theory
   ii. Performing a Draft Check
   iii. Understanding Mold Analysis Settings
   iv. Performing a Thickness Check

Knowledge Check Questions

Module 4. Mold Models
   i. Creating New Mold Models
   ii. Analyzing Model Accuracy
   iii. Locating the Reference Model
   iv. Assembling the Reference Model
   v. Creating the Reference Model
   vi. Redefining the Reference Model
   vii. Analyzing Reference Model Orientation
   viii. Analyzing Mold Cavity Layout
   ix. Analyzing Variable Mold Cavity Layout
   x. Analyzing Mold Cavity Layout Orientation
   xi. Calculating Projected Area

Knowledge Check Questions

Module 5. Shrinkage
   i. Understanding Shrinkage
   ii. Applying Shrinkage by Scale
   iii. Applying Shrinkage by Dimension

Knowledge Check Questions

Module 6. Workpieces
   i. Creating Style States Using the View Manager
   ii. Creating a Workpiece Automatically
Module 7. Mold Volume Creation

i. Surfacing Terms
ii. Understanding Mold Volumes
iii. Sketching Mold Volumes
iv. Creating Sliders using Boundary Quilts
v. Sketching Slider Mold Volumes
vi. Creating a Reference Part Cutout
vii. Sketching Lifter Mold Volumes
viii. Replacing Surfaces and Trimming to Geometry
ix. Sketching Insert Mold Volumes

Knowledge Check Questions

Module 8. Parting Line and Parting Surface Creation

i. Understanding Parting Lines and Parting Surfaces
ii. Creating an Automatic Parting Line Using Silhouette Curves
iii. Analyzing Silhouette Curve Options: Slides
iv. Analyzing Silhouette Curve Options: Loop Selection
v. Creating a Skirt Surface
vi. Analyzing Skirt Surface Options: Extend Curves
vii. Analyzing Skirt Surface Options: Tangent Conditions
viii. Analyzing Skirt Surface Options: Extension Directions
ix. Analyzing Skirt Surface Options: ShutOff Extension
x. Analyzing Surface Editing and Manipulation Tools
xi. Merging Surfaces
xii. Creating Saddle Shutoff Surfaces
xiii. Creating a Parting Surface Manually

Knowledge Check Questions

Module 9. Splitting Mold Volumes

i. Splitting the Workpiece
ii. Splitting Mold Volumes
iii. Splitting Volumes using Multiple Parting Surfaces
iv. Blanking and Unblanking Mold Items
v. Analyzing Split Classification

Knowledge Check Questions

Module 10. Mold Component Extraction

i. Extracting Mold Components from Volumes
ii. Applying Start Models to Mold Components

Knowledge Check Questions
Knowledge Check Questions

Module 11. Mold Features Creation
i. Creating Waterline Circuits
ii. Analyzing Waterline End Conditions
iii. Performing a Waterlines Check
iv. Creating Sprues and Runners
v. Creating Ejector Pin Clearance Holes
vi. Creating UDFs
vii. Placing UDFs

Knowledge Check Questions

Module 12. Filling and Opening the Mold
i. Creating a Molding
ii. Opening the Mold
iii. Draft Checking a Mold Opening Step
iv. Interference Checking a Mold Opening Step
v. Viewing Mold Information

Knowledge Check Questions
Interactive Surface Design using Creo Parametric 2.0

Overview

Course Code: TRN-3921-T  
Course Length: 2 Days

In this course, you will learn how to use the Style environment to create and manipulate freeform curves, freeform surfaces, freeform surface details, and advanced freeform surface models. You will also learn how to integrate style features with other parametric features in design models. After completing this course, you will be well prepared to design complex-shaped freeform surface models in Creo Parametric.

In Creo Parametric, you can create freeform surface models using the Style modeling environment. The Style modeling environment is a spline-based freeform modeler that enables you to combine the parametric feature-based modeling approach with the unconstrained freeform surface modeling approach. This gives you the flexibility to design complex-shaped products in a single modeling environment. The Style tool is available through the Interactive Surface Design Extension (ISDX).

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 style surface modeling process
- Understand style surface modeling concepts
- Create initial style curves
- Develop style surface models
- Use advanced tools and techniques for defining style shapes
- Create smooth style surface models
- Integrate style and parametric features
- Use techniques for creating common detailed shapes
- Create complex, high quality style surface models
Prerequisites

• Introduction to Creo Parametric 2.0

Audience

• This course is intended for design engineers and mechanical designers who need to create styled surface geometry. People in related roles will also benefit from taking this course.
# Agenda

## Day 1

<table>
<thead>
<tr>
<th>Module</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Introduction to the Style Surface Modeling Process</td>
</tr>
<tr>
<td>2</td>
<td>Understanding Style Surface Modeling Concepts</td>
</tr>
<tr>
<td>3</td>
<td>Creating Initial Style Curves</td>
</tr>
<tr>
<td>4</td>
<td>Developing Style Surface Models</td>
</tr>
</tbody>
</table>

## Day 2

<table>
<thead>
<tr>
<th>Module</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>5</td>
<td>Advanced Tools and Techniques for Defining Style Shapes</td>
</tr>
<tr>
<td>6</td>
<td>Creating Smooth Style Surface Models</td>
</tr>
<tr>
<td>7</td>
<td>Integrating Style and Parametric Features</td>
</tr>
<tr>
<td>8</td>
<td>Techniques for Creating Common Detailed Shapes</td>
</tr>
<tr>
<td>9</td>
<td>Creating Complex, High Quality Style Surface Models</td>
</tr>
</tbody>
</table>
Course Content

Module 1. Introduction to the Style Surface Modeling Process
   i. Introduction to Freeform Surface Modeling
   ii. Understanding Surface Modeling Paradigms
   iii. Combining Style and Parametric Modeling
   iv. Typical Style Modeling Process

Knowledge Check Questions

Module 2. Understanding Style Surface Modeling Concepts
   i. Understanding the Style Tool
   ii. Understanding Style Features
   iii. Understanding Datum Features within Style
   iv. Understanding the Style Modeling Environment
   v. Using Style Tool Shortcut Menus
   vi. Using Style Tool Key Combinations
   vii. Understanding Active Planes
   viii. Understanding the Style Tool 4-View Layout
   ix. Understanding Style Preferences

Knowledge Check Questions

Module 3. Creating Initial Style Curves
   i. Understanding Style Curves
   ii. Creating Basic Style Curves
   iii. Creating Style Curves as Circles or Arcs
   iv. Manipulating Style Point Locations
   v. Connecting Curves Using Soft Points
   vi. Manipulating Soft Points
   vii. Defining Endpoint Tangency
   viii. Defining Soft Endpoint Tangency
   ix. Editing Curves
   x. Creating Radial Path Planar Curves
   xi. Using the 4-View Layout to Modify Curves
   xii. Analyzing Curves

Knowledge Check Questions

Module 4. Developing Style Surface Models
   i. Understanding Style Surfaces
   ii. Creating Boundary Surfaces
   iii. Creating Loft Surfaces
   iv. Using Multiple Curves as a Single Boundary
   v. Creating a Blend Surface Using the Radial Option
   vi. Creating a Blend Surface Using the Uniform Option
vii. Using Surfaces to Define Curves
viii. Creating a Curve on Surface Type Curve
ix. Intersecting to Create a Curve on Surface
x. Creating a Curve Using Curve from Surface
xi. Manipulating Curve on Surface Type Curves
xii. Trimming Surfaces in the Style Tool

Knowledge Check Questions

Module 5. Advanced Tools and Techniques for Defining Style Shapes
i. Manipulating Shapes Using Internal Curves
ii. Copying and Moving Curves
iii. Copying Curves Proportionally
iv. Offsetting Curves
v. Modifying Curve Shapes Proportionally
vi. Unlinking Style Curves
vii. Making Curves Planar Between Endpoints
viii. Editing Style Surfaces
ix. Resolving Failed Style Geometry
x. Using References from Design Models
xi. Using Imported 3-D Data

Knowledge Check Questions

Module 6. Creating Smooth Style Surface Models
i. Understanding Curvature
ii. Using the Curvature Analysis Tool
iii. Using Visual Mirror with a Curvature Analysis
iv. Manipulating Surface Boundary Connections
v. Understanding Surface Connection Order
vi. Analyzing Continuity of Style Designs
vii. Using the Shaded Curvature Analysis Tool
viii. Using the Reflection Analysis Tool
ix. Using the Dihedral Angle Analysis Tool

Knowledge Check Questions

Module 7. Integrating Style and Parametric Features
i. Understanding Parallel Modeling
ii. Using Style Surfaces to Define Solid Geometry
iii. Exporting Curve Parameters for Modification
iv. Referencing a Parametric Framework
v. Manipulating Style Geometry Using Editing Tools

Knowledge Check Questions

Module 8. Techniques for Creating Common Detailed Shapes
i. Creating Common Detailed Shapes
ii. Creating Scoops or Bulges Using Intersecting Surfaces
iii. Creating Scoops or Bulges with Definite Boundaries
iv. Creating Scoops or Bulges with Blurred Boundaries
v. Creating Split Surface Geometry

Knowledge Check Questions

Module 9. Creating Complex, High Quality Style Surface Models
i. Understanding Singularity in Triangular Surfaces
ii. Reparameterizing a Surface
iii. Understanding Four-Boundary Surfaces
iv. Using the Overbuild Technique
v. Using the Create Boundaries Technique
vi. Using the Void Boundary Technique
vii. Creating a Four-Boundary Rounded Shape
viii. Creating a Triangular Shape Using Four Boundaries

Knowledge Check Questions
Behavioral Modeling Using Creo Parametric 2.0

Overview

Course Code: TRN-3922-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. After completing this course, you will be prepared to work on critical component designs using Creo Parametric Behavioral Modeling.

In this course, you will focus on learning 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.

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

Course Objectives

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

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

Audience

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

### Day 1

<table>
<thead>
<tr>
<th>Module</th>
<th>1</th>
<th>Introduction to the Behavioral Modeling Process</th>
</tr>
</thead>
<tbody>
<tr>
<td>Module</td>
<td>2</td>
<td>Creating Measurement Features on Creo Parametric Models</td>
</tr>
<tr>
<td>Module</td>
<td>3</td>
<td>Creating Model Property Features on Creo Parametric Models</td>
</tr>
<tr>
<td>Module</td>
<td>4</td>
<td>Creating Analysis Features on Creo Parametric Models</td>
</tr>
<tr>
<td>Module</td>
<td>5</td>
<td>Creating User-Defined Analysis Features on Creo Parametric Models</td>
</tr>
<tr>
<td>Module</td>
<td>6</td>
<td>Conducting Design Studies and Optimizing Models</td>
</tr>
<tr>
<td>Module</td>
<td>7</td>
<td>Project</td>
</tr>
</tbody>
</table>
Course Content

Module 1. Introduction to the Behavioral Modeling Process
   i. Behavioral Modeling Process
   ii. Identifying BMX Analysis Types
   iii. Identifying the Differences Between Creo Parametric Analyses

Knowledge Check Questions

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

Knowledge Check Questions

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

Knowledge Check Questions

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

Knowledge Check Questions

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

Knowledge Check Questions

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

Module 7. Project
Mechanism Design using Creo Parametric 2.0

Overview

Course Code: TRN-3923-T
Course Length: 1 Day

This is used as Course Overview. Course overview and Course Descriptions are same

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 Pro/FICIENCY intended to evaluate your understanding of the course as a whole.

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.
## Agenda

### Day 1

<table>
<thead>
<tr>
<th>Module</th>
<th>1</th>
<th>Introduction to the Mechanism Design Process</th>
</tr>
</thead>
<tbody>
<tr>
<td>Module</td>
<td>2</td>
<td>Creating Mechanism Connections</td>
</tr>
<tr>
<td>Module</td>
<td>3</td>
<td>Configuring Motion and Analysis</td>
</tr>
<tr>
<td>Module</td>
<td>4</td>
<td>Evaluating Analysis Results</td>
</tr>
</tbody>
</table>
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

Knowledge Check Questions

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

Knowledge Check Questions

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

Knowledge Check Questions

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

Knowledge Check Questions
Mechanism Simulation using Creo Parametric 2.0

Overview

Course Code: TRN-3924-T
Course Length: 1 Day

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 experienced users 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 Pro/FICIENCY intended to evaluate your understanding of the course as a whole.

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.
# Agenda

## Day 1

<table>
<thead>
<tr>
<th>Module</th>
<th>1</th>
<th>Introduction to the Mechanism Simulation Process</th>
</tr>
</thead>
<tbody>
<tr>
<td>Module</td>
<td>2</td>
<td>Adding Dynamic Entities to a Mechanism</td>
</tr>
<tr>
<td>Module</td>
<td>3</td>
<td>Analyzing the Mechanism Model</td>
</tr>
<tr>
<td>Module</td>
<td>4</td>
<td>Evaluating Analysis Results</td>
</tr>
<tr>
<td>Module</td>
<td>5</td>
<td>Project</td>
</tr>
</tbody>
</table>
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 Dynamic Analyses
   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
Overview

Course Code: TRN-3925-T  
Course Length: 3 Days

In this course, you will learn how to machine products using Creo Parametric 2.0 manufacturing tools. This course covers creating toolpaths for two axis turning 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 turning sequences, holemaking sequences, and post-process cutter location (CL) data to create machine code. After completing this course, you will be able to create numerical control (NC) programs for turning 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 turning sequences
• Create and modify holemaking sequences
• Use the process manager to create NC sequences
• Post-process cutter location (CL) data
Prerequisites

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

Audience

- This course is intended for manufacturing engineers and NC machinists. People in related roles will also benefit from taking this course.
# Agenda

## Day 1

<table>
<thead>
<tr>
<th>Module</th>
<th>1</th>
<th>Introduction to Turning Manufacturing</th>
</tr>
</thead>
<tbody>
<tr>
<td>Module</td>
<td>2</td>
<td>Creating Turning Manufacturing Models</td>
</tr>
<tr>
<td>Module</td>
<td>3</td>
<td>Configuring Turning Operations</td>
</tr>
<tr>
<td>Module</td>
<td>4</td>
<td>Using Turning Reference Models</td>
</tr>
<tr>
<td>Module</td>
<td>5</td>
<td>Using Turning Workpiece Models</td>
</tr>
<tr>
<td>Module</td>
<td>6</td>
<td>Creating and Using Turning NC Model Assemblies</td>
</tr>
<tr>
<td>Module</td>
<td>7</td>
<td>Creating and Configuring Turning Workcells</td>
</tr>
</tbody>
</table>

## Day 2

<table>
<thead>
<tr>
<th>Module</th>
<th>8</th>
<th>Creating and Configuring Turning Tools</th>
</tr>
</thead>
<tbody>
<tr>
<td>Module</td>
<td>9</td>
<td>Using Template Manufacturing Turning Models</td>
</tr>
<tr>
<td>Module</td>
<td>10</td>
<td>Using Turning Manufacturing Parameters</td>
</tr>
<tr>
<td>Module</td>
<td>11</td>
<td>Creating Area Removal Sequences</td>
</tr>
<tr>
<td>Module</td>
<td>12</td>
<td>Creating Profile Turning Sequences</td>
</tr>
<tr>
<td>Module</td>
<td>13</td>
<td>Creating Grooving Sequences</td>
</tr>
</tbody>
</table>

## Day 3

<table>
<thead>
<tr>
<th>Module</th>
<th>14</th>
<th>Creating Turning Thread Sequences</th>
</tr>
</thead>
<tbody>
<tr>
<td>Module</td>
<td>15</td>
<td>Creating Turning Holemaking Sequences</td>
</tr>
<tr>
<td>Module</td>
<td>16</td>
<td>Using the Process Manager for Turning</td>
</tr>
<tr>
<td>Module</td>
<td>17</td>
<td>Creating and Post-Processing Turning CL Data Files</td>
</tr>
<tr>
<td>Module</td>
<td>18</td>
<td>Turning Project: Needle Valve</td>
</tr>
</tbody>
</table>
Course Content

Module 1. Introduction to Turning Manufacturing
   i. Manufacturing Turning Process Overview
   ii. Understanding the Turning Coordinate System
   iii. Understanding Turning Material Removal
Knowledge Check Questions

Module 2. Creating Turning Manufacturing Models
   i. Creating Turning Manufacturing Models
Knowledge Check Questions

Module 3. Configuring Turning Operations
   i. Configuring Turning Operations
Knowledge Check Questions

Module 4. Using Turning Reference Models
   i. Using Turning Reference Models
Knowledge Check Questions

Module 5. Using Turning Workpiece Models
   i. Using Turning Workpiece Models
Knowledge Check Questions

Module 6. Creating and Using Turning NC Model Assemblies
   i. Creating and Using Turning NC Model Assemblies
Knowledge Check Questions

Module 7. Creating and Configuring Turning Workcells
   i. Creating and Configuring Turning Workcells
Knowledge Check Questions

Module 8. Creating and Configuring Turning Tools
   i. Understanding Turning Tools
   ii. Understanding Turning Tool Orientation
   iii. Creating Standard Turning Tools
   iv. Creating Solid Model Turning Tools
   v. Creating and Using Turning Tool Cutting Data
   vi. Retrieving Turning Tool Data
Knowledge Check Questions

Module 9. Using Template Manufacturing Turning Models
   i. Using Template Manufacturing Turning Models
Knowledge Check Questions

Module 10. Using Turning Manufacturing Parameters
   i. Understanding Turning Manufacturing Parameter Concepts
ii. Configuring Turning Parameter Values
iii. Using Turning Site Parameter Files

Knowledge Check Questions

Module 11. Creating Area Removal Sequences
i. Creating Turning Cut Geometry
ii. Outside Area Removal
iii. Face Area
iv. Inside Area

Knowledge Check Questions

Module 12. Creating Profile Turning Sequences
i. Creating Outside Profile Turning Sequences
ii. Creating Face Profiling Turning Sequences
iii. Creating Inside Profile Turning Sequences

Knowledge Check Questions

Module 13. Creating Grooving Sequences
i. Creating Outside Groove Sequences
ii. Creating Inside Groove Sequences
iii. Creating Face Groove Sequences

Knowledge Check Questions

Module 14. Creating Turning Thread Sequences
i. Creating Outside Thread Sequences
ii. Creating Inside Thread Sequences
iii. Creating Face Thread Sequences

Knowledge Check Questions

Module 15. Creating Turning Holemaking Sequences
i. Creating Drilling Sequences
ii. Creating Countersink Sequences
iii. Creating Tapping Sequences

Knowledge Check Questions

Module 16. Using the Process Manager for Turning
i. Using Process Manager Tools for Turning
ii. Editing Turning Process Items
iii. Creating New Items in the Process Manager for Turning

Knowledge Check Questions

Module 17. Creating and Post-Processing Turning CL Data Files
i. Creating and Post-Processing Turning CL Data Files

Knowledge Check Questions

Module 18. Turning Project: Needle Valve
i. Turning Project: Needle Valve
Manufacturing Update for Creo Parametric 2.0 from Creo Elements/Pro 5.0

Overview

Course Code: TRN-3927–T
Course Length: 1 Day

In this course, you will learn about the manufacturing enhancements made since Creo Elements/Pro 5.0.
First, you will learn about the various enhancements relating to the ribbon-based user interface in manufacturing. Toolpath and CL Data Display Enhancements will then be presented, such as dynamically displaying and updating a toolpath as it is being configured.
You will then study enhancements for cutline milling, trajectory milling, chamfer/round milling and thread milling sequences.
Next, you will learn about the various Volume and Roughing enhancements, such as automatic adjustments in the diameter for helical plunging during roughing and enhancements to parameters to setting equal z-slices during roughing. You will then learn about creating rest finishing sequences, finishing enhancements and creating an in-process stock model. Finally, process manager and other miscellaneous enhancements will be examined.
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

- Utilize enhancements to the manufacturing related ribbon based user interface
- Utilize the enhancements to Toolpath display and CL Data display
- Utilize the enhancements to Cutline Milling, Trajectory Milling, Volume Milling, Roughing, and Finishing
- Create Chamfer and Round Milling sequences
- Create Thread Milling sequences
- Create Rest Finishing sequences
- Create in process stock models
- Utilize the enhancements to the Process Manager
Prerequisites

• Milling using Creo Elements/Pro 5.0 or equivalent experience

Audience

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

**Day 1**

<table>
<thead>
<tr>
<th>Module</th>
<th>1</th>
<th>Ribbon-Based User Interface Enhancements</th>
</tr>
</thead>
<tbody>
<tr>
<td>Module</td>
<td>2</td>
<td>Toolpath and CL Data Display Enhancements</td>
</tr>
<tr>
<td>Module</td>
<td>3</td>
<td>Cutline Enhancements</td>
</tr>
<tr>
<td>Module</td>
<td>4</td>
<td>Trajectory Sequences</td>
</tr>
<tr>
<td>Module</td>
<td>5</td>
<td>Creating Chamfer and Round Milling Sequences</td>
</tr>
<tr>
<td>Module</td>
<td>6</td>
<td>Creating Thread Milling Sequences</td>
</tr>
<tr>
<td>Module</td>
<td>7</td>
<td>Volume and Roughing Enhancements</td>
</tr>
<tr>
<td>Module</td>
<td>8</td>
<td>Creating Rest Finishing Sequences</td>
</tr>
<tr>
<td>Module</td>
<td>9</td>
<td>Finishing Enhancements</td>
</tr>
<tr>
<td>Module</td>
<td>10</td>
<td>Creating In-Process Stock Model</td>
</tr>
<tr>
<td>Module</td>
<td>11</td>
<td>Process Manager Enhancements</td>
</tr>
<tr>
<td>Module</td>
<td>12</td>
<td>Miscellaneous Enhancements</td>
</tr>
</tbody>
</table>
Course Content

Module 1. Ribbon-Based User Interface Enhancements
   i. Ribbon-Based User Interface
   ii. Engraving on Flat and Complex Surfaces
   iii. Cutline Option
   iv. Arc Connections in Face Milling

Knowledge Check Questions

Module 2. Toolpath and CL Data Display Enhancements
   i. Dynamic Preview for Toolpath Display
   ii. Display Tool Preview
   iii. CL Data Window

Knowledge Check Questions

Module 3. Cutline Enhancements
   i. Auto Cutline Creation
   ii. Circular Output while Machining Complex Holes
   iii. Cutline Milling
   iv. Projected Toolpath for 4–5-Axis Cutline

Knowledge Check Questions

Module 4. Trajectory Sequences
   i. Start Point for Closed Trajectory Loops
   ii. Start-End Points for Open Trajectory Toolpaths
   iii. Helical 3-Axis Trajectory

Knowledge Check Questions

Module 5. Creating Chamfer and Round Milling Sequences
   i. Creating Chamfer Milling Sequences
   ii. Creating Round Milling Sequences

Knowledge Check Questions

Module 6. Creating Thread Milling Sequences
   i. Creating Thread Milling Sequences

Knowledge Check Questions

Module 7. Volume and Roughing Enhancements
   i. Helical Plunge Automatic Adjustment
   ii. Ramp Motion Optimization
   iii. Equal Space Z-Slices
   iv. Variable S-Connections

Knowledge Check Questions

Module 8. Creating Rest Finishing Sequences
   i. Creating Rest Finishing Sequences
Knowledge Check Questions

Module 9. Finishing Enhancements
   i. Minimize Sharp Corners in Finishing
   ii. Additional Bottom Clearance for Profile Portion
   iii. Optimize Approach and Exit for Profile Moves

Knowledge Check Questions

Module 10. Creating In-Process Stock Model
   i. Creating In-Process Stock Model

Knowledge Check Questions

Module 11. Process Manager Enhancements
   i. Process Manager Enhancements

Knowledge Check Questions

Module 12. Miscellaneous Enhancements
   i. Open Tool Motions
   ii. Drill Group Definition

Knowledge Check Questions
Advanced Turning and Multi-Task Machining using Creo Parametric 2.0

Overview

Course Code: TRN-3930-T
Course Length: 2 Days

In this course, you will learn how to machine components using mill-turn machines, twin-turret machines, and in-line twin spindle lathes. You will also learn how to create turning tool paths and milling tool paths for each machine type, where appropriate. In addition, you will understand how to post-process mill-turn toolpaths and how to use the NC-Check and Vericut material removal simulation tools. Finally, you will learn how to view machine tool simulations for 3-axis milling, 2-axis turning, and 5-axis mill-turn machines.

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

Course Objectives

- Learn the basic mill-turn process
- Create mill-turn work centers
- Create various milling and drilling toolpaths using live tooling
- Learn the basic twin turret turning process
- Create twin turret turning work centers
- Create toolpaths for twin turret lathes
- Learn the basic in-line twin spindle turning process
- Create in-line twin spindle lathe work centers
- Create toolpaths using in-line spindles
- Configure and use multi-tip tools
- Understand mill-turn post processing
- Learn how to use material removal simulation
- Learn how to use machine tool simulation
Prerequisites

• Milling using Creo Parametric 2.0 or Turning using Creo Parametric 2.0

Audience

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

## Day 1

<table>
<thead>
<tr>
<th>Module</th>
<th>1</th>
<th>Mill-Turn Machining</th>
</tr>
</thead>
<tbody>
<tr>
<td>Module</td>
<td>2</td>
<td>Twin Turret Turning</td>
</tr>
</tbody>
</table>

## Day 2

<table>
<thead>
<tr>
<th>Module</th>
<th>3</th>
<th>In-Line Twin Spindle Turning</th>
</tr>
</thead>
<tbody>
<tr>
<td>Module</td>
<td>4</td>
<td>Configuring and Using Multi-Tip Tools</td>
</tr>
<tr>
<td>Module</td>
<td>5</td>
<td>Mill-Turn Post Processing</td>
</tr>
<tr>
<td>Module</td>
<td>6</td>
<td>Material Removal Simulation</td>
</tr>
<tr>
<td>Module</td>
<td>7</td>
<td>Machine Tool Simulation</td>
</tr>
</tbody>
</table>
Course Content

Module 1. Mill-Turn Machining
   i. Mill-Turn Process
   ii. Creating Mill-Turn Work Centers
   iii. Creating 3-Axis Milling and Drilling Toolpaths Using Live Tooling
   iv. Creating 4-Axis Milling and Drilling Toolpaths Using Live Tooling
   v. Creating 5-Axis Milling and Drilling Toolpaths Using Live Tooling

Knowledge Check Questions

Module 2. Twin Turret Turning
   i. Twin Turret Turning Process
   ii. Creating Twin Turret Turning Work Centers
   iii. Creating Toolpaths for Twin Turret Lathes

Knowledge Check Questions

Module 3. In-Line Twin Spindle Turning
   i. In-Line Twin Spindle Turning Process
   ii. Creating In-Line Twin Spindle Lathe Work Centers
   iii. Creating Toolpaths with In-Line Spindles

Knowledge Check Questions

Module 4. Configuring and Using Multi-Tip Tools
   i. Configuring and Using Multi-Tip Tools

Knowledge Check Questions

Module 5. Mill-Turn Post Processing
   i. Mill-Turn Post Processing

Knowledge Check Questions

Module 6. Material Removal Simulation
   i. Material Removal Simulation

Knowledge Check Questions

Module 7. Machine Tool Simulation
   i. 2-Axis Lathe Machine Tool Simulation
   ii. 3-Axis Mill Machine Tool Simulation
   iii. 5-Axis Mill-Turn Machine Tool Simulation

Knowledge Check Questions
Creo Parametric 2.0 for SolidWorks Users

Overview

<table>
<thead>
<tr>
<th>Course Code</th>
<th>TRN-4000-T</th>
</tr>
</thead>
<tbody>
<tr>
<td>Course Length</td>
<td>3 Days</td>
</tr>
</tbody>
</table>

In this course, you will learn about core modeling skills. This comprehensive, hands-on course is specifically designed for existing SolidWorks users who want to become proficient with Creo Parametric as quickly as possible. Topics include understanding the interface and basic Creo Parametric concepts, selecting and editing, sketching tools, and basic feature, part, and assembly creation.

The course also includes a comprehensive design project that enables you to practice your new skills by creating realistic parts. After completing the course, you will be well prepared to work effectively on product design projects using 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

- Learn the basic Creo Parametric modeling process
- 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 extrudes, revolves, and profile ribs
- Utilize internal sketches
- Create sweeps and blends
- Create holes, shells, and drafts
- Create rounds and chamfers
- Assemble with constraints
- Lay out drawings and create views
- Create drawing annotations
- Investigate parent/child relationships
- Resolve failures and seek help
- Complete a comprehensive design project
Prerequisites

• Knowledge of SolidWorks, including an understanding of solid modeling, feature-based, parametric, and associative concepts

Audience

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

### Day 1

<table>
<thead>
<tr>
<th>Module</th>
<th>1</th>
<th>Introduction to the Creo Parametric Basic Modeling Process</th>
</tr>
</thead>
<tbody>
<tr>
<td>Module</td>
<td>2</td>
<td>Using the Creo Parametric Interface</td>
</tr>
<tr>
<td>Module</td>
<td>3</td>
<td>Editing Geometry, Features, and Models</td>
</tr>
<tr>
<td>Module</td>
<td>4</td>
<td>Sketcher Geometry and Tools</td>
</tr>
</tbody>
</table>

### Day 2

<table>
<thead>
<tr>
<th>Module</th>
<th>5</th>
<th>Creating Extrudes, Revolves, and Ribs</th>
</tr>
</thead>
<tbody>
<tr>
<td>Module</td>
<td>6</td>
<td>Creating Sweeps and Blends</td>
</tr>
<tr>
<td>Module</td>
<td>7</td>
<td>Creating Holes, Shells, and Draft</td>
</tr>
<tr>
<td>Module</td>
<td>8</td>
<td>Creating Rounds and Chamfers</td>
</tr>
</tbody>
</table>

### Day 3

<table>
<thead>
<tr>
<th>Module</th>
<th>9</th>
<th>Project I</th>
</tr>
</thead>
<tbody>
<tr>
<td>Module</td>
<td>10</td>
<td>Assembling with Constraints</td>
</tr>
<tr>
<td>Module</td>
<td>11</td>
<td>Introduction to Drawings</td>
</tr>
<tr>
<td>Module</td>
<td>12</td>
<td>Resolving Failures and Seeking Help</td>
</tr>
</tbody>
</table>
Course Content

Module 1. Introduction to the Creo Parametric Basic Modeling Process
   i. Creo Parametric Basic Modeling Process

Module 2. Using the Creo Parametric Interface
   i. Understanding the Main Interface
   ii. Setting the Working Directory and Opening and Saving Files
   iii. Understanding the Ribbon Interface
   iv. Customizing the Ribbon Interface
   v. Analyzing Basic 3-D Orientation
   vi. Creating and Managing View Orientations
   vii. Setting Up New Part Models
   viii. Understanding Creo Parametric Basic Controls
   ix. Understanding the Model Tree
   x. Selecting Items using Direct Selection
   xi. Selecting Items using Query Selection
   xii. Using the Search Tool
   xiii. Using the Smart Selection Filter
   xiv. Understanding Selection Filters

Knowledge Check Questions

Module 3. Editing Geometry, Features, and Models
   i. Using Drag Handles and Dimension Draggers
   ii. Understanding Regeneration and Auto Regeneration
   iii. Editing Features
   iv. Editing Features using Edit Definition
   v. Activating and Editing Models
   vi. Deleting and Suppressing Items
   vii. Editing Feature and Component Visibility
   viii. Applying Shape Selection
   ix. Applying Flexible Move using the Dragger
   x. Applying Flexible Move by Dimension

Knowledge Check Questions

Module 4. Sketcher Geometry and Tools
   i. Reviewing Sketcher Theory
   ii. Modifying the Sketcher Display
   iii. Utilizing Constraints
   iv. Sketching with On-the-Fly Constraints
   v. Sketching Lines
   vi. Sketching Rectangles and Parallelograms
   vii. Sketching Circles
viii. Using Geometry Tools within Sketcher
ix. Manipulating Sketches within Sketcher
ox. Dimensioning Entities within Sketcher
xi. Modifying Dimensions within Sketcher
xii. Sketcher Conflicts
xiii. Creating Sketches (‘Sketch’ Feature)
xiv. Specifying and Manipulating the Sketch Setup
xv. Utilizing Sketch References

Knowledge Check Questions

Module 5. 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. Automatically Adding and Removing Material
   ix. Creating Profile Rib Features
   x. Creating Internal Sketches

Knowledge Check Questions

Module 6. 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 7. Creating Holes, Shells, and Draft
   i. Common Dashboard Options Hole Depth
   ii. Creating Coaxial Holes
   iii. Creating Linear Holes
   iv. Creating Shell Features
   v. Creating Draft Features
   vi. Creating Basic Split Drafts

Knowledge Check Questions

Module 8. 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 Full Rounds  
v. Creating Round Sets  
vi. Creating Chamfer Sets

Knowledge Check Questions

Module 9. Project I
i. The Air Circulator  
ii. Piston Assembly Components  
iii. Crankshaft, Engine Block, Impeller, and Impeller Housing  
iv. The Frame and Bolt

Module 10. 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. Assembling using Automatic

Knowledge Check Questions

Module 11. Introduction to Drawings
i. Drawing Development Process  
ii. Analyzing Drawing Concepts and Theory  
iii. Analyzing Basic 2-D Orientation  
iv. Creating New Drawings and Applying Formats  
v. Creating and Orienting General Views  
vi. Adding Drawing Models  
vii. Creating Projection Views  
viii. Analyzing Annotation Concepts and Types  
ix. Showing, Erasing, and Deleting Annotations  
x. Cleaning Up Dimensions  
xi. Manipulating Dimensions

Knowledge Check Questions

Module 12. Resolving Failures and Seeking Help
i. Understanding Parent Child Relationships  
ii. Viewing Part Parent Child Information  
iii. Reordering Features  
iv. Inserting Features  
v. Understanding and Identifying Failures  
vi. Analyzing Geometry Failures
vii. Analyzing Missing Part Reference Failures
viii. Using Creo Parametric Help

Knowledge Check Questions
Web Based Curriculum Guide

- Update to Creo Parametric 2.0 from Creo Elements/Pro 5.0
- Update to Creo Parametric 2.0 from Pro/ENGINEER Wildfire 4.0
- Introduction to Creo Parametric 2.0 - Fundamentals
- Introduction to Creo Parametric 2.0 - Productivity Tools
- Advanced Modeling using Creo Parametric 2.0
- Advanced Assembly Design using Creo Parametric 2.0
- Detailing using Creo Parametric 2.0
- Surfacing using Creo Parametric 2.0
- Sheetmetal Design using Creo Parametric 2.0
- Milling using Creo Parametric 2.0
- Cabling using Creo Parametric 2.0
- Piping using Creo Parametric 2.0
- Introduction to Creo Simulate 2.0
- Introduction to Creo Schematics 2.0
- Introduction to Creo Layout 2.0
- Introduction to Creo Illustrate 2.0
- Flexible Modeling using Creo Parametric 2.0
• Introduction to Creo Direct 2.0
• Introduction to Creo Options Modeler 2.0
• Mold Design using Creo Parametric 2.0
• Interactive Surface Design using Creo Parametric 2.0
• Behavioral Modeling using Creo Parametric 2.0
• Mechanism Design using Creo Parametric 2.0
• Mechanism Simulation using Creo Parametric 2.0
• Turning using Creo Parametric 2.0
• Cabling Update for Creo Parametric 2.0 from Pro/ENGINEER Wildfire 4.0
• Manufacturing Update for Creo Parametric 2.0 from Creo Elements/Pro 5.0
• Update to Creo Simulate 2.0 from Pro/ENGINEER Wildfire 4.0
• Update to Creo Schematics 2.0 from Routed Systems Designer 9.0
• Advanced Turning and Multi-task Machining using Creo Parametric 2.0
• Manikin using Creo Parametric 2.0
• Freestyle Surface Design using Creo Parametric 2.0
• Creo Parametric 2.0 for SolidWorks Users
Update to Creo Parametric 2.0 from Creo Elements/Pro 5.0

Overview

Course Code: WBT-3900-0
Course Length: 8 Hours

In this course, you will learn how to utilize the core functionality enhancements in Creo Parametric 2.0. First, you will become familiar with using and customizing the new ribbon interface in Creo Parametric. The new measure and sectioning interfaces will also be examined. Next, you will become familiar with the Sketcher workflow and reference enhancements. Part modeling enhancements to features such as Extrude, Corner Chamfer, Sweeps, Blends, and Datum Curves will then be examined. You will also learn about new and enhanced Assembly capabilities, such as selecting multiple components, the new relationship constraints, and enhancements for dragging components. Then, you will examine the new Table and Balloon functionality for 2-D drawings, and review various detailing enhancements. Finally, in Sheetmetal mode you will learn to use the many updated Wall, Bend, and Relief tools, as well as the consolidated Flat Pattern tool and configuring Sheetmetal properties.

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

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

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

Audience

• This course is intended for design engineers, mechanical designers, and industrial designers
• People in related roles can also benefit from taking this course
# Table of Contents

<table>
<thead>
<tr>
<th>Module</th>
<th>1</th>
<th>Interface Enhancements</th>
</tr>
</thead>
<tbody>
<tr>
<td>Module</td>
<td>2</td>
<td>Sketcher Enhancements</td>
</tr>
<tr>
<td>Module</td>
<td>3</td>
<td>Part Modeling Enhancements</td>
</tr>
<tr>
<td>Module</td>
<td>4</td>
<td>Assembly Enhancements</td>
</tr>
<tr>
<td>Module</td>
<td>5</td>
<td>Drawing Enhancements</td>
</tr>
<tr>
<td>Module</td>
<td>6</td>
<td>Sheetmetal Enhancements</td>
</tr>
<tr>
<td>Module</td>
<td>7</td>
<td>Appendix</td>
</tr>
</tbody>
</table>

Overview

In this course, you will learn how to utilize the core functionality enhancements in Creo Parametric 2.0. First, you will become familiar with using and customizing the new ribbon interface in Creo Parametric. The new measure and sectioning interfaces will also be examined. Next, you will study the Sketcher workflow and reference enhancements, as well as Sketcher constraint, geometry, and diagnostics enhancements. Part modeling enhancements to features such as Extrude, Trajectory Rib, Point Pattern, Corner Chamfer, Sweeps, Blends, UDFs, and Datum Curves will then be examined. You will also learn about new and enhanced Assembly capabilities such as selecting multiple components, the new relationship constraints, enhancements for dragging components, explode enhancements, simplified rep enhancements, and dynamic gear enhancements. Then you will learn how to identify and resolve part and assembly failures on the fly, without entering Resolve mode. Next, you will examine the new Drawing tree and drawing sheets tabs, as well as showing annotations. Also in Drawing mode, you will learn the new Table and Balloon functionality for 2-D drawings, and review various detailing enhancements. Finally, in Sheetmetal mode you will learn to use the updated Wall, Bend, and Relief tools as well as the consolidated Flat Pattern tool and configuring Sheetmetal properties.

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

- Utilize the Interface enhancements in Creo Parametric 2.0
- Utilize the Sketcher enhancements in Creo Parametric 2.0
- Utilize the Modeling enhancements in Creo Parametric 2.0
- Utilize the Assembly enhancements in Creo Parametric 2.0
- Utilize the Drawing enhancements in Creo Parametric 2.0
- Utilize the Sheetmetal enhancements in Creo Parametric 2.0

Prerequisites

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

Audience

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

<table>
<thead>
<tr>
<th>Module</th>
<th>1</th>
<th>Interface Enhancements</th>
</tr>
</thead>
<tbody>
<tr>
<td>Module</td>
<td>2</td>
<td>Sketcher Enhancements</td>
</tr>
<tr>
<td>Module</td>
<td>3</td>
<td>Part Modeling Enhancements</td>
</tr>
<tr>
<td>Module</td>
<td>4</td>
<td>Assembly Enhancements</td>
</tr>
<tr>
<td>Module</td>
<td>5</td>
<td>Resolving Failures</td>
</tr>
<tr>
<td>Module</td>
<td>6</td>
<td>Drawing Enhancements</td>
</tr>
<tr>
<td>Module</td>
<td>7</td>
<td>Sheetmetal Enhancements</td>
</tr>
<tr>
<td>Module</td>
<td>8</td>
<td>Appendix</td>
</tr>
</tbody>
</table>
Introduction to Creo Parametric 2.0 - Fundamentals

Overview

Course Code       WBT-3902-A

Course Length     20 Hours

This course is designed for new users who want to become proficient with Creo Parametric as quickly as possible. You will focus on learning core-modeling skills in this comprehensive, hands-on course. Topics include understanding the interface and basic Creo Parametric concepts, selecting and editing, sketching and sketcher tools, and basic feature creation. The course also includes a comprehensive design project that enables you to practice your new skills by creating realistic parts. After completing the course you will be well-prepared to work effectively on product design projects using Creo Parametric.

Course Objectives

• Learning the basic Creo Parametric modeling process
• Understanding Creo Parametric concepts
• Learning how to use the Creo Parametric interface
• Selecting and editing geometry, features, and models
• Sketching geometry and using tools
• Creating sketches for features
• Creating datum planes and datum axes
• Creating extrudes, revolves, and profile ribs
• Utilizing internal sketches and embedded datums
• Creating sweeps and blends
• Creating holes, shells, and drafts
• Creating rounds and chamfers
• Comprehensive Design Project

Prerequisites

• None

Audience

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

<table>
<thead>
<tr>
<th>Module</th>
<th>1</th>
<th>Introduction to the Creo Parametric Basic Modeling Process</th>
</tr>
</thead>
<tbody>
<tr>
<td>Module</td>
<td>2</td>
<td>Understanding Creo Parametric Concepts</td>
</tr>
<tr>
<td>Module</td>
<td>3</td>
<td>Using the Creo Parametric Interface</td>
</tr>
<tr>
<td>Module</td>
<td>4</td>
<td>Selecting Geometry, Features, and Models</td>
</tr>
<tr>
<td>Module</td>
<td>5</td>
<td>Editing Geometry, Features, and Models</td>
</tr>
<tr>
<td>Module</td>
<td>6</td>
<td>Creating Sketcher Geometry</td>
</tr>
<tr>
<td>Module</td>
<td>7</td>
<td>Using Sketcher Tools</td>
</tr>
<tr>
<td>Module</td>
<td>8</td>
<td>Creating Sketches for Features</td>
</tr>
<tr>
<td>Module</td>
<td>9</td>
<td>Creating Datum Features: Planes and Axes</td>
</tr>
<tr>
<td>Module</td>
<td>10</td>
<td>Creating Extrudes, Revolves, and Ribs</td>
</tr>
<tr>
<td>Module</td>
<td>11</td>
<td>Utilizing Internal Sketches and Embedded Datums</td>
</tr>
<tr>
<td>Module</td>
<td>12</td>
<td>Creating Sweeps and Blends</td>
</tr>
<tr>
<td>Module</td>
<td>13</td>
<td>Creating Holes, Shells, and Draft</td>
</tr>
<tr>
<td>Module</td>
<td>14</td>
<td>Creating Rounds and Chamfers</td>
</tr>
<tr>
<td>Module</td>
<td>15</td>
<td>Project I</td>
</tr>
</tbody>
</table>
Introduction to Creo Parametric 2.0 - Productivity Tools

Overview

This course is designed for existing Creo Parametric users who want to increase their productivity. You will focus on learning advanced modeling skills in this comprehensive, hands-on course. Topics include patterning; measuring and inspecting models; group, copy, and mirror tools; assembly creation; explode states and explode line creation; drawing creation; using layers; investigating parent/child relationships; capturing design intent; and resolving failures and seeking help. 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.

Course Objectives

- Grouping, copying, and mirroring items
- Creating patterns
- Measuring and inspecting models
- Assembling with constraints
- Assembling with connections
- Exploding assemblies
- Laying out drawings and creating views
- Creating drawing annotations
- Using layers
- Investigating parent/child relationships
- Capturing and managing design intent
- Resolving failures and seeking help
- Comprehensive Design Project

Prerequisites

- Introduction to Creo Parametric - Fundamentals

Audience

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

<table>
<thead>
<tr>
<th>Module</th>
<th>1</th>
<th>Group, Copy, and Mirror Tools</th>
</tr>
</thead>
<tbody>
<tr>
<td>Module</td>
<td>2</td>
<td>Creating Patterns</td>
</tr>
<tr>
<td>Module</td>
<td>3</td>
<td>Measuring and Inspecting Models</td>
</tr>
<tr>
<td>Module</td>
<td>4</td>
<td>Assembling with Constraints</td>
</tr>
<tr>
<td>Module</td>
<td>5</td>
<td>Assembling with Connections</td>
</tr>
<tr>
<td>Module</td>
<td>6</td>
<td>Exploding Assemblies</td>
</tr>
<tr>
<td>Module</td>
<td>7</td>
<td>Drawing Layout and Views</td>
</tr>
<tr>
<td>Module</td>
<td>8</td>
<td>Creating Drawing Annotations</td>
</tr>
<tr>
<td>Module</td>
<td>9</td>
<td>Using Layers</td>
</tr>
<tr>
<td>Module</td>
<td>10</td>
<td>Investigating Parent/Child Relationships</td>
</tr>
<tr>
<td>Module</td>
<td>11</td>
<td>Capturing and Managing Design Intent</td>
</tr>
<tr>
<td>Module</td>
<td>12</td>
<td>Resolving Failures and Seeking Help</td>
</tr>
<tr>
<td>Module</td>
<td>13</td>
<td>Project II</td>
</tr>
</tbody>
</table>
Advanced Modeling using Creo Parametric 2.0

Overview

Course Code: WBT-3903-0
Course Length: 24 Hours

The Advanced Modeling using Creo Parametric 2.0 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 2.0.

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 2.0
- Update to Creo Parametric 2.0 from Creo Elements/Pro 5.0
Audience

- This course is intended for mechanical designers, design engineers. People in related roles will also benefit from taking this course.
<table>
<thead>
<tr>
<th>Module</th>
<th>1</th>
<th>Advanced Selection</th>
</tr>
</thead>
<tbody>
<tr>
<td>Module</td>
<td>2</td>
<td>Advanced Datum Features</td>
</tr>
<tr>
<td>Module</td>
<td>3</td>
<td>Advanced Sketching</td>
</tr>
<tr>
<td>Module</td>
<td>4</td>
<td>Advanced Hole Creation</td>
</tr>
<tr>
<td>Module</td>
<td>5</td>
<td>Advanced Drafts and Ribs</td>
</tr>
<tr>
<td>Module</td>
<td>6</td>
<td>Advanced Shells</td>
</tr>
<tr>
<td>Module</td>
<td>7</td>
<td>Advanced Rounds and Chamfers</td>
</tr>
<tr>
<td>Module</td>
<td>8</td>
<td>Relations and Parameters</td>
</tr>
<tr>
<td>Module</td>
<td>9</td>
<td>Advanced Blends</td>
</tr>
<tr>
<td>Module</td>
<td>10</td>
<td>Sweeps with Variable Sections</td>
</tr>
<tr>
<td>Module</td>
<td>11</td>
<td>Helical Sweeps</td>
</tr>
<tr>
<td>Module</td>
<td>12</td>
<td>Swept Blends</td>
</tr>
<tr>
<td>Module</td>
<td>13</td>
<td>Advanced Layers</td>
</tr>
<tr>
<td>Module</td>
<td>14</td>
<td>Advanced Reference Management</td>
</tr>
<tr>
<td>Module</td>
<td>15</td>
<td>Family Tables</td>
</tr>
<tr>
<td>Module</td>
<td>16</td>
<td>Reusing Features</td>
</tr>
<tr>
<td>Module</td>
<td>17</td>
<td>Advanced Copy</td>
</tr>
<tr>
<td>Module</td>
<td>18</td>
<td>Advanced Patterns</td>
</tr>
</tbody>
</table>
Advanced Assembly Design using Creo Parametric 2.0

Overview

Course Code: WBT-3904-0
Course Length: 24 Hours

In this course, you will learn how to use Creo Parametric 2.0 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 assembly constraints
- Create and using component interfaces
- Create and using 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, layer states, and combined views
- Substitute components using user-defined reps, envelopes, and simplified reps
- Understand advanced simplified rep functionality
- Create and use assembly structure and skeletons
Prerequisites

- Introduction to Creo Parametric 2.0
- Update to Creo Parametric 2.0 from Creo Elements/Pro 5.0

Audience

- This course is intended for design engineers, mechanical designers, and related roles
# Table of Contents

<table>
<thead>
<tr>
<th>Module</th>
<th>1</th>
<th>Using Advanced Assembly Constraints</th>
</tr>
</thead>
<tbody>
<tr>
<td>Module</td>
<td>2</td>
<td>Creating and Using Component Interfaces</td>
</tr>
<tr>
<td>Module</td>
<td>3</td>
<td>Creating and Using Flexible Components</td>
</tr>
<tr>
<td>Module</td>
<td>4</td>
<td>Restructuring and Mirroring Assemblies</td>
</tr>
<tr>
<td>Module</td>
<td>5</td>
<td>Using Assembly Features and Shrinkwrap</td>
</tr>
<tr>
<td>Module</td>
<td>6</td>
<td>Replacing Components in an Assembly</td>
</tr>
<tr>
<td>Module</td>
<td>7</td>
<td>Understanding the Basics of Simplified Reps</td>
</tr>
<tr>
<td>Module</td>
<td>8</td>
<td>Creating Cross-Sections, Display Styles, Layer States, and Combined Views</td>
</tr>
<tr>
<td>Module</td>
<td>9</td>
<td>Substituting Components using User Defined, Envelopes, and Simplified Reps</td>
</tr>
<tr>
<td>Module</td>
<td>10</td>
<td>Understanding Advanced Simplified Rep Functionality</td>
</tr>
<tr>
<td>Module</td>
<td>11</td>
<td>Creating and Using Assembly Structure and Skeletons</td>
</tr>
<tr>
<td>Module</td>
<td>12</td>
<td>Project</td>
</tr>
</tbody>
</table>
Overview

<table>
<thead>
<tr>
<th>Course Code</th>
<th>WBT-3905-0</th>
</tr>
</thead>
<tbody>
<tr>
<td>Course Length</td>
<td>24 Hours</td>
</tr>
</tbody>
</table>

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 2.0 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 2.0

Audience

• This course is intended for mechanical designers and design engineers. People in related roles will also benefit from taking this course.
## Table of Contents

<table>
<thead>
<tr>
<th>Module</th>
<th>1</th>
<th>Introduction to Drawings</th>
</tr>
</thead>
<tbody>
<tr>
<td>Module</td>
<td>2</td>
<td>Creating New Drawings</td>
</tr>
<tr>
<td>Module</td>
<td>3</td>
<td>Creating Drawing Views</td>
</tr>
<tr>
<td>Module</td>
<td>4</td>
<td>Adding Model Details to Drawings</td>
</tr>
<tr>
<td>Module</td>
<td>5</td>
<td>Adding Notes to Drawings</td>
</tr>
<tr>
<td>Module</td>
<td>6</td>
<td>Adding Tolerance Information</td>
</tr>
<tr>
<td>Module</td>
<td>7</td>
<td>Adding Draft Geometry and Symbols</td>
</tr>
<tr>
<td>Module</td>
<td>8</td>
<td>Using Layers in Drawings</td>
</tr>
<tr>
<td>Module</td>
<td>9</td>
<td>Creating and Using Tables in Drawings</td>
</tr>
<tr>
<td>Module</td>
<td>10</td>
<td>Using Report Information in Drawings</td>
</tr>
<tr>
<td>Module</td>
<td>11</td>
<td>Creating Drawing Formats</td>
</tr>
<tr>
<td>Module</td>
<td>12</td>
<td>Configuring the Drawing Environment</td>
</tr>
<tr>
<td>Module</td>
<td>13</td>
<td>Managing Large Drawings</td>
</tr>
</tbody>
</table>
Surfacing using Creo Parametric 2.0

Overview

Course Code: WBT-3906-0
Course Length: 24 Hours

In this course, you will learn how to use various techniques to create complex surfaces with tangent and curvature continuities. You will also learn how 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 2.0
• Update to Creo Parametric 2.0 from Creo Elements/Pro 5.0

Audience

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

<table>
<thead>
<tr>
<th>Module</th>
<th>1</th>
<th>Surface Modeling Overview</th>
</tr>
</thead>
<tbody>
<tr>
<td>Module</td>
<td>2</td>
<td>Advanced Selection</td>
</tr>
<tr>
<td>Module</td>
<td>3</td>
<td>Advanced Datum Features</td>
</tr>
<tr>
<td>Module</td>
<td>4</td>
<td>Advanced Sketching</td>
</tr>
<tr>
<td>Module</td>
<td>5</td>
<td>Basic Surfacing Tools</td>
</tr>
<tr>
<td>Module</td>
<td>6</td>
<td>Boundary Blend Surfaces</td>
</tr>
<tr>
<td>Module</td>
<td>7</td>
<td>Sweep Surfaces with Variable Sections</td>
</tr>
<tr>
<td>Module</td>
<td>8</td>
<td>Helical Sweeps</td>
</tr>
<tr>
<td>Module</td>
<td>9</td>
<td>Swept Blends</td>
</tr>
<tr>
<td>Module</td>
<td>10</td>
<td>Analyzing Surface Curvature</td>
</tr>
<tr>
<td>Module</td>
<td>11</td>
<td>Additional Surface Analysis Tools</td>
</tr>
<tr>
<td>Module</td>
<td>12</td>
<td>Extending and Trimming Surfaces</td>
</tr>
<tr>
<td>Module</td>
<td>13</td>
<td>Manipulating Surfaces</td>
</tr>
<tr>
<td>Module</td>
<td>14</td>
<td>Creating and Editing Solids using Quilts</td>
</tr>
<tr>
<td>Module</td>
<td>15</td>
<td>Master Model Technique</td>
</tr>
<tr>
<td>Module</td>
<td>16</td>
<td>Project</td>
</tr>
</tbody>
</table>
Sheetmetal Design using Creo Parametric 2.0

Overview

<table>
<thead>
<tr>
<th>Course Code</th>
<th>WBT-3907-0</th>
</tr>
</thead>
<tbody>
<tr>
<td>Course Length</td>
<td>16 Hours</td>
</tr>
</tbody>
</table>

Sheetmetal Design using Creo Parametric 2.0 is a comprehensive training course that teaches you how to create sheetmetal parts in Creo Parametric. The course builds upon the basic lessons you learned in Introduction to Creo Parametric 2.0 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

- The sheetmetal design process
- Sheetmetal model creation, conversion, and display
- Methods of developed length calculation
- Primary wall features
- Secondary wall features
- Partial walls
- Bend relief
- Unbend and bend back features
- Sheetmetal bend features
- Flat patterns
- Sheetmetal cuts
- Forms
- Notch and punch features
- Sheetmetal environment setup
- Sheetmetal design Information tools
- Sheetmetal design rules
- Detailing sheetmetal designs
- Sheetmetal design project
Prerequisites

- Introduction to Creo Parametric 2.0

Audience

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

<table>
<thead>
<tr>
<th>Module</th>
<th>1</th>
<th>Introduction to the Creo Parametric Sheetmetal Design Process</th>
</tr>
</thead>
<tbody>
<tr>
<td>Module</td>
<td>2</td>
<td>Sheetmetal Model Fundamentals</td>
</tr>
<tr>
<td>Module</td>
<td>3</td>
<td>Creating Primary Sheetmetal Wall Features</td>
</tr>
<tr>
<td>Module</td>
<td>4</td>
<td>Creating Secondary Sheetmetal Wall Features</td>
</tr>
<tr>
<td>Module</td>
<td>5</td>
<td>Bending and Unbending Sheetmetal Models</td>
</tr>
<tr>
<td>Module</td>
<td>6</td>
<td>Modifying Sheetmetal Models</td>
</tr>
<tr>
<td>Module</td>
<td>7</td>
<td>Sheetmetal Setup and Tools</td>
</tr>
<tr>
<td>Module</td>
<td>8</td>
<td>Detailing Sheetmetal Designs</td>
</tr>
<tr>
<td>Module</td>
<td>9</td>
<td>Design Project</td>
</tr>
</tbody>
</table>
Milling using Creo Parametric 2.0

Overview

<table>
<thead>
<tr>
<th>Course Code</th>
<th>WBT-3908-0</th>
</tr>
</thead>
<tbody>
<tr>
<td>Course Length</td>
<td>40 Hours</td>
</tr>
</tbody>
</table>

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

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

Course Objectives

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

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

Audience

• This course is intended for manufacturing engineers and NC machinists
## Table of Contents

<table>
<thead>
<tr>
<th>Module</th>
<th>1</th>
<th>Introduction to Manufacturing</th>
</tr>
</thead>
<tbody>
<tr>
<td>Module</td>
<td>2</td>
<td>Creating Manufacturing Models</td>
</tr>
<tr>
<td>Module</td>
<td>3</td>
<td>Configuring Operations</td>
</tr>
<tr>
<td>Module</td>
<td>4</td>
<td>Using Reference Models</td>
</tr>
<tr>
<td>Module</td>
<td>5</td>
<td>Using Workpiece Models</td>
</tr>
<tr>
<td>Module</td>
<td>6</td>
<td>Creating and Using NC Model Assemblies</td>
</tr>
<tr>
<td>Module</td>
<td>7</td>
<td>Creating and Configuring Workcells</td>
</tr>
<tr>
<td>Module</td>
<td>8</td>
<td>Creating and Configuring Tools</td>
</tr>
<tr>
<td>Module</td>
<td>9</td>
<td>Using Template Manufacturing Models</td>
</tr>
<tr>
<td>Module</td>
<td>10</td>
<td>Using Manufacturing Parameters</td>
</tr>
<tr>
<td>Module</td>
<td>11</td>
<td>Creating Face Milling Sequences</td>
</tr>
<tr>
<td>Module</td>
<td>12</td>
<td>Creating Volume Milling Sequences</td>
</tr>
<tr>
<td>Module</td>
<td>13</td>
<td>Creating Profile Milling Sequences</td>
</tr>
<tr>
<td>Module</td>
<td>14</td>
<td>Creating Straight Cut Surface Milling Sequences</td>
</tr>
<tr>
<td>Module</td>
<td>15</td>
<td>Creating From Surface Isolines Surface Milling Sequences</td>
</tr>
<tr>
<td>Module</td>
<td>16</td>
<td>Creating Cut Line Surface Milling Sequences</td>
</tr>
<tr>
<td>Module</td>
<td>17</td>
<td>Advanced Surface Milling Options</td>
</tr>
<tr>
<td>Module</td>
<td>18</td>
<td>Creating Roughing and Re-roughing Sequences</td>
</tr>
<tr>
<td>Module</td>
<td>19</td>
<td>Creating Finishing Sequences</td>
</tr>
<tr>
<td>Module</td>
<td>20</td>
<td>Creating Trajectory Milling Sequences</td>
</tr>
<tr>
<td>Module</td>
<td>21</td>
<td>Creating Holemaking Sequences</td>
</tr>
<tr>
<td>Module</td>
<td>22</td>
<td>Creating Engraving Sequences</td>
</tr>
<tr>
<td>Module</td>
<td>23</td>
<td>Using the Process Manager</td>
</tr>
<tr>
<td>Module</td>
<td>24</td>
<td>Creating and Post-Processing CL Data Files</td>
</tr>
</tbody>
</table>
Cabling using Creo Parametric 2.0

Overview

<table>
<thead>
<tr>
<th>Course Code</th>
<th>WBT-3909-0</th>
</tr>
</thead>
<tbody>
<tr>
<td>Course Length</td>
<td>24 Hours</td>
</tr>
</tbody>
</table>

In this course, you will learn how to create 3-D electrical harnesses using Creo Parametric 2.0. This includes using Creo Schematics to pass schematic diagram information into the 3-D harness designs created within Creo Parametric 2.0. 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 2.0. 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 2.0.

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

Course Objectives

- Learn the basic Creo Parametric 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 flat harness
• Document harness designs
• Comprehensive design project

Prerequisites

• Introduction to Creo Parametric 2.0 or equivalent experience
• Introduction to Creo Schematics 2.0 (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.
<table>
<thead>
<tr>
<th>Module</th>
<th>1</th>
<th>Introduction to the Creo Basic Cabling Process</th>
</tr>
</thead>
<tbody>
<tr>
<td>Module</td>
<td>2</td>
<td>Creating Harness Assembly Structures</td>
</tr>
<tr>
<td>Module</td>
<td>3</td>
<td>Setting Up for Cabling</td>
</tr>
<tr>
<td>Module</td>
<td>4</td>
<td>Routing Wires and Cables</td>
</tr>
<tr>
<td>Module</td>
<td>5</td>
<td>Modifying Wire Routings</td>
</tr>
<tr>
<td>Module</td>
<td>6</td>
<td>Routing and Utilizing Networks</td>
</tr>
<tr>
<td>Module</td>
<td>7</td>
<td>Establishing Logical References</td>
</tr>
<tr>
<td>Module</td>
<td>8</td>
<td>Routing Wires and Cables using Logical Data</td>
</tr>
<tr>
<td>Module</td>
<td>9</td>
<td>Creating Harness Components and Cosmetics</td>
</tr>
<tr>
<td>Module</td>
<td>10</td>
<td>Creating Flat Harnesses</td>
</tr>
<tr>
<td>Module</td>
<td>11</td>
<td>Documenting Harness Designs</td>
</tr>
<tr>
<td>Module</td>
<td>12</td>
<td>Project (Creo Schematics-Based)</td>
</tr>
<tr>
<td>Module</td>
<td>13</td>
<td>Project (Manual Routing)</td>
</tr>
</tbody>
</table>
Overview

In this course, you will learn how to manually create (non-specification driven) mechanical piping designs using Creo Parametric 2.0. This includes learning how to configure pipelines, how to route pipelines, and how to insert pipe fittings such as valves and reducers. You will also learn how to create specification driven industrial piping designs using Creo Parametric 2.0. This includes learning how to use schematic diagrams created with Creo Schematics to drive 3-D industrial piping designs created within Creo Parametric 2.0. Finally, you learn how to document piping designs by creating drawings that include BOM tables, pipe bend tables, and engineering information, as well as how to export ISOGEN format files for creating pipeline, spool and systems isometric drawings.

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

Course Objectives

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

- Introduction to Creo Parametric 2.0 or equivalent experience

Audience

- This course is intended for engineers who are involved in the 3-D routing of mechanical piping systems and industrial piping systems. People in related roles will also benefit from taking this course.
<table>
<thead>
<tr>
<th>Module</th>
<th>1</th>
<th>Introduction to Piping</th>
</tr>
</thead>
<tbody>
<tr>
<td>Module</td>
<td>2</td>
<td>Creating Piping Assembly Structures</td>
</tr>
<tr>
<td>Module</td>
<td>3</td>
<td>Configuring and Routing Pipelines</td>
</tr>
<tr>
<td>Module</td>
<td>4</td>
<td>Moving and Modifying Pipelines</td>
</tr>
<tr>
<td>Module</td>
<td>5</td>
<td>Configuring and Inserting Fittings</td>
</tr>
<tr>
<td>Module</td>
<td>6</td>
<td>Creating Solid Pipeline Models</td>
</tr>
<tr>
<td>Module</td>
<td>7</td>
<td>Gathering Piping Information</td>
</tr>
<tr>
<td>Module</td>
<td>8</td>
<td>Creating Piping Drawings</td>
</tr>
<tr>
<td>Module</td>
<td>9</td>
<td>Specification Database Overview</td>
</tr>
<tr>
<td>Module</td>
<td>10</td>
<td>Setting Up Specification Databases: Piping</td>
</tr>
<tr>
<td>Module</td>
<td>11</td>
<td>Setting Up Specification Databases: Fittings</td>
</tr>
<tr>
<td>Module</td>
<td>12</td>
<td>General Master Catalog Files</td>
</tr>
<tr>
<td>Module</td>
<td>13</td>
<td>Configuring Project-Specific Data Files</td>
</tr>
<tr>
<td>Module</td>
<td>14</td>
<td>Specification-Driven Routing and Inserting Fittings</td>
</tr>
<tr>
<td>Module</td>
<td>15</td>
<td>Using Creo Schematics Process and Instrumentation Diagrams Data</td>
</tr>
<tr>
<td>Module</td>
<td>16</td>
<td>Schematic Driven Pipeline Modeling</td>
</tr>
<tr>
<td>Module</td>
<td>17</td>
<td>Using ISOGEN PCF Data</td>
</tr>
</tbody>
</table>
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

- Learn the basic Simulate analysis process
- Learn theory and simulate model topics
- Explore results
- Learn about 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
Prerequisites

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

Audience

• This course is intended for design engineers and mechanical designers. People in related roles will also benefit from taking this course.
## Table of Contents

<table>
<thead>
<tr>
<th>Module</th>
<th>1</th>
<th>Introduction to Creo Simulate</th>
</tr>
</thead>
<tbody>
<tr>
<td>Module</td>
<td>2</td>
<td>Theoretical Foundations</td>
</tr>
<tr>
<td>Module</td>
<td>3</td>
<td>Simulation Models</td>
</tr>
<tr>
<td>Module</td>
<td>4</td>
<td>Materials and Material Properties</td>
</tr>
<tr>
<td>Module</td>
<td>5</td>
<td>Structural Constraints</td>
</tr>
<tr>
<td>Module</td>
<td>6</td>
<td>Structural Loads</td>
</tr>
<tr>
<td>Module</td>
<td>7</td>
<td>Meshing</td>
</tr>
<tr>
<td>Module</td>
<td>8</td>
<td>Convergence</td>
</tr>
<tr>
<td>Module</td>
<td>9</td>
<td>Structural Analysis</td>
</tr>
<tr>
<td>Module</td>
<td>10</td>
<td>Introduction to Results Evaluation</td>
</tr>
<tr>
<td>Module</td>
<td>11</td>
<td>Refining the Design</td>
</tr>
<tr>
<td>Module</td>
<td>12</td>
<td>Basic Model Debugging</td>
</tr>
<tr>
<td>Module</td>
<td>13</td>
<td>Singularities</td>
</tr>
<tr>
<td>Module</td>
<td>14</td>
<td>Analyzing Assemblies</td>
</tr>
<tr>
<td>Module</td>
<td>15</td>
<td>Shells</td>
</tr>
<tr>
<td>Module</td>
<td>16</td>
<td>Idealizations</td>
</tr>
<tr>
<td>Module</td>
<td>17</td>
<td>Thermal Analysis</td>
</tr>
<tr>
<td>Module</td>
<td>18</td>
<td>Advanced Analysis</td>
</tr>
<tr>
<td>Module</td>
<td>19</td>
<td>Project</td>
</tr>
</tbody>
</table>
Introduction to Creo Schematics 2.0

Overview

Course Code  WBT-3912-0
Course Length  24 Hours

In this course, you will learn how to use Creo Schematics to create schematic diagrams for both electrical harness and piping designs. You will also learn how to administer the Creo Schematics working environment, configure the catalog library, and create functional block diagrams and block interconnect diagrams. In addition, you will learn how to create electrical circuit diagrams, wiring diagrams (including wire interconnect diagrams), and process and instrumentation design diagrams for piping systems. Finally, you will learn how to use wiring diagrams and process and instrumentation design diagrams to configure 3-D harness designs and industrial piping designs created within 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

• Understand the 2-D schematic design processes for electrical harness designs and piping designs
• Configure the Creo Schematics working environment
• Create and configure the catalog library
• Create functional block diagrams (including block interconnect diagrams)
• Create circuit diagrams
• Create wiring diagrams (including wire interconnect diagrams)
• Create process and instrumentation design diagrams
• Communicate diagram information to Creo Parametric
Prerequisites

• None

Audience

• This course is intended for engineers involved in the schematic 2-D layout of either electrical diagrams or piping diagrams
Table of Contents

<table>
<thead>
<tr>
<th>Module</th>
<th>1</th>
<th>Introduction to Creo Schematics</th>
</tr>
</thead>
<tbody>
<tr>
<td>Module</td>
<td>2</td>
<td>Configuring the Working Environment</td>
</tr>
<tr>
<td>Module</td>
<td>3</td>
<td>Configuring Designs and Diagram Sheets</td>
</tr>
<tr>
<td>Module</td>
<td>4</td>
<td>Configuring Catalog Properties</td>
</tr>
<tr>
<td>Module</td>
<td>5</td>
<td>Creating Design Templates and Template Sheets</td>
</tr>
<tr>
<td>Module</td>
<td>6</td>
<td>Using Creo Schematics Tools</td>
</tr>
<tr>
<td>Module</td>
<td>7</td>
<td>Creating Catalog Artifacts</td>
</tr>
<tr>
<td>Module</td>
<td>8</td>
<td>Creating and Configuring a Central Catalog</td>
</tr>
<tr>
<td>Module</td>
<td>9</td>
<td>Reviewing Designs</td>
</tr>
<tr>
<td>Module</td>
<td>10</td>
<td>Instancing and Manipulating Objects</td>
</tr>
<tr>
<td>Module</td>
<td>11</td>
<td>Creating Block Diagrams</td>
</tr>
<tr>
<td>Module</td>
<td>12</td>
<td>Creating Circuit Diagrams</td>
</tr>
<tr>
<td>Module</td>
<td>13</td>
<td>Creating Wiring Diagrams</td>
</tr>
<tr>
<td>Module</td>
<td>14</td>
<td>Creating Interconnect Diagrams</td>
</tr>
<tr>
<td>Module</td>
<td>15</td>
<td>Creating P&amp;ID Diagrams</td>
</tr>
<tr>
<td>Module</td>
<td>16</td>
<td>Using Schematic Design Data in Creo Parametric Harness and Piping Designs</td>
</tr>
<tr>
<td>Module</td>
<td>17</td>
<td>Using Pro/Diagram Information</td>
</tr>
</tbody>
</table>
Introduction to Creo Layout 2.0

Overview

Course Code: WBT-3913-0  
Course Length: 8 Hours

In this course, you will learn about a new Creo module called Layout 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 makes use of precision panels and sketching guides to intelligently sketch a variety of 2-D geometry, as well as learn how to manipulate, organize, and import layout geometry. Finally, you will learn how to leverage 2-D Layout designs in 3-D models.

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

Course Objectives

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

• Introduction to Creo Parametric 2.0 (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.
# Table of Contents

<table>
<thead>
<tr>
<th>Module</th>
<th>1</th>
<th>Introduction to Layout</th>
</tr>
</thead>
<tbody>
<tr>
<td>Module</td>
<td>2</td>
<td>Layout Sketching Methodology</td>
</tr>
<tr>
<td>Module</td>
<td>3</td>
<td>Sketching Layout Geometry</td>
</tr>
<tr>
<td>Module</td>
<td>4</td>
<td>Manipulating Layout Geometry</td>
</tr>
<tr>
<td>Module</td>
<td>5</td>
<td>Organizing Layout Geometry</td>
</tr>
<tr>
<td>Module</td>
<td>6</td>
<td>Importing Layout Geometry</td>
</tr>
<tr>
<td>Module</td>
<td>7</td>
<td>Annotating Layouts</td>
</tr>
<tr>
<td>Module</td>
<td>8</td>
<td>Utilizing Layouts in 3-D Models</td>
</tr>
</tbody>
</table>
Introduction to Creo Illustrate 2.0

Overview

<table>
<thead>
<tr>
<th>Course Code</th>
<th>WBT-3914-0</th>
</tr>
</thead>
<tbody>
<tr>
<td>Course Length</td>
<td>8 Hours</td>
</tr>
</tbody>
</table>

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

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

Course Objectives

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

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

**Audience**

- This course is intended for technical publications illustrators, technical publications authors, training authors, manufacturing instructors, and users responsible for parts definition, technical marketing, and service planning. People in related roles will also benefit from taking this course.
<table>
<thead>
<tr>
<th>Module</th>
<th>1</th>
<th>Introduction to Creo Illustrate</th>
</tr>
</thead>
<tbody>
<tr>
<td>Module</td>
<td>2</td>
<td>Create an Illustration and Work with Figures</td>
</tr>
<tr>
<td>Module</td>
<td>3</td>
<td>Exploding Assemblies</td>
</tr>
<tr>
<td>Module</td>
<td>4</td>
<td>Authoring Sequenced Process Steps</td>
</tr>
<tr>
<td>Module</td>
<td>5</td>
<td>Creo Illustrate Animations</td>
</tr>
<tr>
<td>Module</td>
<td>6</td>
<td>Create and Manage Annotations, Sub-Assemblies, and Parts Lists</td>
</tr>
<tr>
<td>Module</td>
<td>7</td>
<td>Publishing and Exporting 3-D Illustrations</td>
</tr>
</tbody>
</table>
Flexible Modeling using Creo Parametric 2.0

Overview

<table>
<thead>
<tr>
<th>Course Code</th>
<th>WBT-3915-0</th>
</tr>
</thead>
<tbody>
<tr>
<td>Course Length</td>
<td>8 Hours</td>
</tr>
</tbody>
</table>

In this course, you will learn how to use flexible modeling tools to edit existing geometry on parametric models. The flexible modeling process typically involves initially selecting model surfaces, then refining the selected surface set using smart selection tools, and finally modifying the selected geometry by applying transformation tools, patterning tools, or symmetry tools. At the end of each module, you will complete a set of review questions to reinforce critical topics from that module. At the end of the course, you will complete a course assessment in Pro/FICIENCY intended to evaluate your understanding of the course as a whole.

Course Objectives

- Understand flexible modeling basics
- Apply selection and tools
- Utilize editing and transformations
- Work with recognition
- Use propagation and other editing features

Prerequisites

- Introduction to Creo Parametric 2.0 or equivalent experience

Audience

- This course is intended for design engineers, mechanical designers, and industrial designers. People in related roles will also benefit from taking this course.
<table>
<thead>
<tr>
<th>Module</th>
<th>1</th>
<th>Introduction to Flexible Modeling</th>
</tr>
</thead>
<tbody>
<tr>
<td>Module</td>
<td>2</td>
<td>Selection and Tools</td>
</tr>
<tr>
<td>Module</td>
<td>3</td>
<td>Editing and Transformations</td>
</tr>
<tr>
<td>Module</td>
<td>4</td>
<td>Recognition</td>
</tr>
<tr>
<td>Module</td>
<td>5</td>
<td>Propagation and Other Editing Features</td>
</tr>
</tbody>
</table>
Introduction to Creo Direct 2.0

Overview

In this course, you will learn direct modeling using Creo Direct 2.0. 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 by extruding, revolving, and sweeping sketches and sketch regions
- Create datums, holes, rounds, chamfers, and draft
- Use advanced sketching techniques
- Rapidly select and directly modify 3-D shapes in your models
- Use advanced tools including sweeps and shells
- Understand the direct approach to creating assemblies
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.
<table>
<thead>
<tr>
<th>Module</th>
<th>1</th>
<th>Introduction to Creo Direct</th>
</tr>
</thead>
<tbody>
<tr>
<td>Module</td>
<td>2</td>
<td>Creating Sketches in 2-D Mode</td>
</tr>
<tr>
<td>Module</td>
<td>3</td>
<td>Extruding and Revolving Sketches and Sketch Regions</td>
</tr>
<tr>
<td>Module</td>
<td>4</td>
<td>Datums, Holes, Rounds, Chamfers, and Drafts</td>
</tr>
<tr>
<td>Module</td>
<td>5</td>
<td>Advanced Sketching</td>
</tr>
<tr>
<td>Module</td>
<td>6</td>
<td>Selecting, Modifying, and Reusing 3-D Geometry</td>
</tr>
<tr>
<td>Module</td>
<td>7</td>
<td>Advanced Tools: Sweep and Shell</td>
</tr>
<tr>
<td>Module</td>
<td>8</td>
<td>Creating Assemblies in Creo Direct</td>
</tr>
</tbody>
</table>
Introduction to Creo Options Modeler 2.0

Overview

<table>
<thead>
<tr>
<th>Course Code</th>
<th>WBT-3917-0</th>
</tr>
</thead>
<tbody>
<tr>
<td>Course Length</td>
<td>8 Hours</td>
</tr>
</tbody>
</table>

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

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

Course Objectives

- Understand and use Creo Options Modeler
- Create interchange assemblies
- Create configurable modules and products
- Define options and choices
- Utilize configurable assemblies
Prerequisites

- Introduction to Creo Parametric 2.0
- Advanced Assembly Design using Creo Parametric 2.0

Audience

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

<table>
<thead>
<tr>
<th>Module</th>
<th>1</th>
<th>Introduction to Creo Options Modeler</th>
</tr>
</thead>
<tbody>
<tr>
<td>Module</td>
<td>2</td>
<td>Creating Interchange Assemblies</td>
</tr>
<tr>
<td>Module</td>
<td>3</td>
<td>Creating Configurable Modules and Products</td>
</tr>
<tr>
<td>Module</td>
<td>4</td>
<td>Defining Options and Choices</td>
</tr>
<tr>
<td>Module</td>
<td>5</td>
<td>Utilizing Configurable Assemblies</td>
</tr>
<tr>
<td>Module</td>
<td>6</td>
<td>Project</td>
</tr>
</tbody>
</table>
Mold Design using Creo Parametric 2.0

Overview

Course Code: WBT-3920-0
Course Length: 16 Hours

The Mold application provides the tools to create a mold model from start to finish using the mold design process within Creo Parametric 2.0. In this course, you will learn how to create, modify, and analyze mold components and assemblies. Any changes made to the design model automatically propagate to the mold components and assemblies. You will also learn how to create final extract components that reflect the geometry of the design model, along with shrinkage considerations, adequate drafting, mold features, and cooling systems. After completing this course, you will have a better understanding of the mold design process and how to create molded products using the mold design 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 Pro/FICIENCY intended to evaluate your understanding of the course as a whole.

Course Objectives

• Learn the basic mold process
• Prepare design models for the mold process
• Analyze design models to ensure their readiness for molding
• Create mold models
• Apply shrinkage to the reference model
• Create and assemble workpieces into the mold model
• Create mold volumes
• Create parting lines and parting surfaces
• Split mold volumes
• Extract mold components
• Create mold features
• Learn how to fill and open the mold

Prerequisites

• Introduction to Creo Parametric 2.0
• Basic understanding of industry standard Mold design terminology and processes
• Knowledge of Creo Parametric surfacing techniques a plus
Audience

- This course is intended for designers, machinists, and manufacturing engineers. People in related roles will also benefit from taking this course.
<table>
<thead>
<tr>
<th>Module</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Module 1</td>
<td>Introduction to the Creo Parametric Basic Mold Process</td>
</tr>
<tr>
<td>Module 2</td>
<td>Design Model Preparation</td>
</tr>
<tr>
<td>Module 3</td>
<td>Design Model Analysis</td>
</tr>
<tr>
<td>Module 4</td>
<td>Mold Models</td>
</tr>
<tr>
<td>Module 5</td>
<td>Shrinkage</td>
</tr>
<tr>
<td>Module 6</td>
<td>Workpieces</td>
</tr>
<tr>
<td>Module 7</td>
<td>Mold Volume Creation</td>
</tr>
<tr>
<td>Module 8</td>
<td>Parting Line and Parting Surface Creation</td>
</tr>
<tr>
<td>Module 9</td>
<td>Splitting Mold Volumes</td>
</tr>
<tr>
<td>Module 10</td>
<td>Mold Component Extraction</td>
</tr>
<tr>
<td>Module 11</td>
<td>Mold Features Creation</td>
</tr>
<tr>
<td>Module 12</td>
<td>Filling and Opening the Mold</td>
</tr>
</tbody>
</table>
Interactive Surface Design using Creo Parametric 2.0

Overview

Course Code: WBT-3921-0
Course Length: 16 Hours

In this course, you will learn how to use the Style environment to create and manipulate freeform curves, freeform surfaces, freeform surface details, and advanced freeform surface models. You will also learn how to integrate style features with other parametric features in design models. After completing this course, you will be well prepared to design complex-shaped freeform surface models in Creo Parametric.

In Creo Parametric, you can create freeform surface models using the Style modeling environment. The Style modeling environment is a spline-based freeform modeler that enables you to combine the parametric feature-based modeling approach with the unconstrained freeform surface modeling approach. This gives you the flexibility to design complex-shaped products in a single modeling environment. The Style tool is available through the Interactive Surface Design Extension (ISDX).

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 style surface modeling process
- Understand style surface modeling concepts
- Create initial style curves
- Develop style surface models
- Use advanced tools and techniques for defining style shapes
- Create smooth style surface models
- Integrate style and parametric features
- Use techniques for creating common detailed shapes
- Create complex, high quality style surface models
Prerequisites

• Introduction to Creo Parametric 2.0

Audience

• This course is intended for design engineers and mechanical designers who need to create styled surface geometry. People in related roles will also benefit from taking this course.
## Table of Contents

<table>
<thead>
<tr>
<th>Module</th>
<th>1</th>
<th>Introduction to the Style Surface Modeling Process</th>
</tr>
</thead>
<tbody>
<tr>
<td>Module</td>
<td>2</td>
<td>Understanding Style Surface Modeling Concepts</td>
</tr>
<tr>
<td>Module</td>
<td>3</td>
<td>Creating Initial Style Curves</td>
</tr>
<tr>
<td>Module</td>
<td>4</td>
<td>Developing Style Surface Models</td>
</tr>
<tr>
<td>Module</td>
<td>5</td>
<td>Advanced Tools and Techniques for Defining Style Shapes</td>
</tr>
<tr>
<td>Module</td>
<td>6</td>
<td>Creating Smooth Style Surface Models</td>
</tr>
<tr>
<td>Module</td>
<td>7</td>
<td>Integrating Style and Parametric Features</td>
</tr>
<tr>
<td>Module</td>
<td>8</td>
<td>Techniques for Creating Common Detailed Shapes</td>
</tr>
<tr>
<td>Module</td>
<td>9</td>
<td>Creating Complex, High Quality Style Surface Models</td>
</tr>
</tbody>
</table>
Behavioral Modeling Using Creo Parametric 2.0

Overview

Course Code: WBT-3922-0

Course Length: 8 Hours

This course is designed for experienced users who want to add additional features to meet or exceed the design specifications of their products. After completing this course, you will be prepared to work on critical component designs using Creo Parametric Behavioral Modeling.

In this course, you will focus on learning 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.

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

Course Objectives

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

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

Audience

• This course is intended for product designers and engineers. People in related roles will also benefit from taking this course.
<table>
<thead>
<tr>
<th>Module</th>
<th>1</th>
<th>Introduction to the Behavioral Modeling Process</th>
</tr>
</thead>
<tbody>
<tr>
<td>Module</td>
<td>2</td>
<td>Creating Measurement Features on Creo Parametric Models</td>
</tr>
<tr>
<td>Module</td>
<td>3</td>
<td>Creating Model Property Features on Creo Parametric Models</td>
</tr>
<tr>
<td>Module</td>
<td>4</td>
<td>Creating Analysis Features on Creo Parametric Models</td>
</tr>
<tr>
<td>Module</td>
<td>5</td>
<td>Creating User-Defined Analysis Features on Creo Parametric Models</td>
</tr>
<tr>
<td>Module</td>
<td>6</td>
<td>Conducting Design Studies and Optimizing Models</td>
</tr>
<tr>
<td>Module</td>
<td>7</td>
<td>Project</td>
</tr>
</tbody>
</table>
Mechanism Design using Creo Parametric 2.0

Overview

<table>
<thead>
<tr>
<th>Course Code</th>
<th>WBT-3923–0</th>
</tr>
</thead>
<tbody>
<tr>
<td>Course Length</td>
<td>8 Hours</td>
</tr>
</tbody>
</table>

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 Pro/FICIENCY intended to evaluate your understanding of the course as a whole.

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.
Table of Contents

<table>
<thead>
<tr>
<th>Module</th>
<th>1</th>
<th>Introduction to the Mechanism Design Process</th>
</tr>
</thead>
<tbody>
<tr>
<td>Module</td>
<td>2</td>
<td>Creating Mechanism Connections</td>
</tr>
<tr>
<td>Module</td>
<td>3</td>
<td>Configuring Motion and Analysis</td>
</tr>
<tr>
<td>Module</td>
<td>4</td>
<td>Evaluating Analysis Results</td>
</tr>
</tbody>
</table>
Mechanism Simulation using Creo Parametric 2.0

Overview

Course Code: WBT-3924-0
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 experienced users 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 Pro/FICIENCY intended to evaluate your understanding of the course as a whole.

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.
# Table of Contents

<table>
<thead>
<tr>
<th>Module</th>
<th>1</th>
<th>2</th>
<th>3</th>
<th>4</th>
<th>5</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Introduction to the Mechanism Simulation Process</td>
<td>Adding Dynamic Entities to a Mechanism</td>
<td>Analyzing the Mechanism Model</td>
<td>Evaluating Analysis Results</td>
<td>Project</td>
</tr>
</tbody>
</table>
Turning using Creo Parametric 2.0

Overview

Course Code | WBT-3925-0
---|---
Course Length | 24 Hours

In this course, you will learn how to machine products using Creo Parametric 2.0 manufacturing tools. This course covers creating toolpaths for two axis turning 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 turning sequences, holemaking sequences, and post-process cutter location (CL) data to create machine code. After completing this course, you will be able to create numerical control (NC) programs for turning 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 turning sequences
- Create and modify holemaking sequences
- Use the process manager to create NC sequences
- Post-process cutter location (CL) data
Prerequisites

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

Audience

- This course is intended for manufacturing engineers and NC machinists. People in related roles will also benefit from taking this course.
## Table of Contents

<table>
<thead>
<tr>
<th>Module</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Introduction to Turning Manufacturing</td>
</tr>
<tr>
<td>2</td>
<td>Creating Turning Manufacturing Models</td>
</tr>
<tr>
<td>3</td>
<td>Configuring Turning Operations</td>
</tr>
<tr>
<td>4</td>
<td>Using Turning Reference Models</td>
</tr>
<tr>
<td>5</td>
<td>Using Turning Workpiece Models</td>
</tr>
<tr>
<td>6</td>
<td>Creating and Using Turning NC Model Assemblies</td>
</tr>
<tr>
<td>7</td>
<td>Creating and Configuring Turning Workcells</td>
</tr>
<tr>
<td>8</td>
<td>Creating and Configuring Turning Tools</td>
</tr>
<tr>
<td>9</td>
<td>Using Template Manufacturing Turning Models</td>
</tr>
<tr>
<td>10</td>
<td>Using Turning Manufacturing Parameters</td>
</tr>
<tr>
<td>11</td>
<td>Creating Area Removal Sequences</td>
</tr>
<tr>
<td>12</td>
<td>Creating Profile Turning Sequences</td>
</tr>
<tr>
<td>13</td>
<td>Creating Grooving Sequences</td>
</tr>
<tr>
<td>14</td>
<td>Creating Turning Thread Sequences</td>
</tr>
<tr>
<td>15</td>
<td>Creating Turning Holemaking Sequences</td>
</tr>
<tr>
<td>16</td>
<td>Using the Process Manager for Turning</td>
</tr>
<tr>
<td>17</td>
<td>Creating and Post-Processing Turning CL Data Files</td>
</tr>
<tr>
<td>18</td>
<td>Turning Project: Needle Valve</td>
</tr>
</tbody>
</table>
Cabling Update for Creo Parametric 2.0 from Pro/ENGINEER Wildfire 4.0

Overview

Course Code  | WBT-3926-0
Course Length | 6 Hours

In this course, you will learn about the enhancements made to Cabling mode in Creo Parametric 2.0.

First, you will explore and customize the Cabling ribbon interface. Enhancements with component designation will then be presented, such as manually designating connectors, designating with entry port parameters, and designating components on-the-fly. You will then study enhancements when importing Creo Schematics data and auto-designating missing components. Next, you will learn about the various wire routing enhancements, such as using simple route, inserting and editing locations, following cables, rerouting wires, and modifying wire lengths. Finally, routing with networks will be examined, both with and without the use of logical data.

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

Course Objectives

• Understand the Cabling Interface and Setup Enhancements
• Understand Manual Routing Enhancements
• Understand Network and Logical Routing Enhancements
Prerequisites

- Introduction to Creo Parametric 2.0 or equivalent experience
- Introduction to Creo Schematics 2.0 (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.
# Table of Contents

<table>
<thead>
<tr>
<th>Module</th>
<th>1</th>
<th>Cabling Interface and Setup Enhancements</th>
</tr>
</thead>
<tbody>
<tr>
<td>Module</td>
<td>2</td>
<td>Manual Routing Enhancements</td>
</tr>
<tr>
<td>Module</td>
<td>3</td>
<td>Network and Logical Routing Enhancements</td>
</tr>
</tbody>
</table>
Manufacturing Update for Creo Parametric 2.0 from Creo Elements/Pro 5.0

Overview

Course Code: WBT–3927–0
Course Length: 8 Hours

In this course, you will learn about the manufacturing enhancements made since Creo Elements/Pro 5.0. First, you will learn about the various enhancements relating to the ribbon-based user interface in manufacturing. Toolpath and CL Data Display Enhancements will then be presented, such as dynamically displaying and updating a toolpath as it is being configured. You will then study enhancements for cutline milling, trajectory milling, chamfer/round milling and thread milling sequences.

Next, you will learn about the various Volume and Roughing enhancements, such as automatic adjustments in the diameter for helical plunging during roughing and enhancements to parameters to setting equal z-slices during roughing. You will then learn about creating rest finishing sequences, finishing enhancements and creating an in-process stock model. Finally, process manager and other miscellaneous enhancements will be examined.

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

- Utilize enhancements to the manufacturing related ribbon based user interface
- Utilize the enhancements to Toolpath display and CL Data display
- Utilize the enhancements to Cutline Milling, Trajectory Milling, Volume Milling, Roughing, and Finishing
- Create Chamfer and Round Milling sequences
- Create Thread Milling sequences
- Create Rest Finishing sequences
- Create in process stock models
- Utilize the enhancements to the Process Manager
Prerequisites

- Milling using Creo Elements/Pro 5.0 or equivalent experience

Audience

- This course is intended for manufacturing engineers and NC machinists. People in related roles will also benefit from taking this course.
<table>
<thead>
<tr>
<th>Module</th>
<th>1</th>
<th>Ribbon-Based User Interface Enhancements</th>
</tr>
</thead>
<tbody>
<tr>
<td>Module</td>
<td>2</td>
<td>Toolpath and CL Data Display Enhancements</td>
</tr>
<tr>
<td>Module</td>
<td>3</td>
<td>Cutline Enhancements</td>
</tr>
<tr>
<td>Module</td>
<td>4</td>
<td>Trajectory Sequences</td>
</tr>
<tr>
<td>Module</td>
<td>5</td>
<td>Creating Chamfer and Round Milling Sequences</td>
</tr>
<tr>
<td>Module</td>
<td>6</td>
<td>Creating Thread Milling Sequences</td>
</tr>
<tr>
<td>Module</td>
<td>7</td>
<td>Volume and Roughing Enhancements</td>
</tr>
<tr>
<td>Module</td>
<td>8</td>
<td>Creating Rest Finishing Sequences</td>
</tr>
<tr>
<td>Module</td>
<td>9</td>
<td>Finishing Enhancements</td>
</tr>
<tr>
<td>Module</td>
<td>10</td>
<td>Creating In-Process Stock Model</td>
</tr>
<tr>
<td>Module</td>
<td>11</td>
<td>Process Manager Enhancements</td>
</tr>
<tr>
<td>Module</td>
<td>12</td>
<td>Miscellaneous Enhancements</td>
</tr>
</tbody>
</table>
Overview

In this course, you will learn how to utilize the core functionality enhancements in Creo Simulate 2.0. This course reviews the enhancements introduced from Pro/ENGINEER Wildfire 4.0 through Creo Simulate 2.0. First, you will become familiar with using the new standalone Creo Simulate application. The new ribbon interface will also be examined. Next, you will study the Creo Simulate workflow, diagnostics, and units enhancements, as well as the new process guide template editor. Analysis enhancements such as contact combined with large deformation, plasticity with large strain, load histories for nonlinear static analyses, and batch run processes will then be examined. You will also learn about new and enhanced result evaluation capabilities, such as default templates and measure versus measure graphs in results. You will then review the new AutoGEM settings and controls. Next, you will learn how to apply a volumetric bolt preload, review a new shell pair definition workflow, work with shells with variable thickness, and apply an improved base excitation model. Finally, in Thermal mode, you will learn how to create heat loads, apply traveling heat loads, and apply a temperature load to a Creo Simulate structure model.

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

- Identify the enhancements from Pro/ENGINEER Wildfire Mechanica 4.0 through Creo Simulate 2.0
- Use the Creo Simulate 2.0 interface
- Understand the units and diagnostic enhancements
- Use the batch run process
- Use load histories for nonlinear static analysis
- Understand the result window options
- Understand the new AutoGEM controls and settings
- Understand the new shell capabilities
- Apply base excitation to a structure
- Create a heat load
- Apply a traveling heat load
- Apply a temperature load to a structure model

Prerequisites

- Three months of Pro/ENGINEER Wildfire 4.0 or 5.0, or Creo Parametric experience
- Three months of Pro/ENGINEER Wildfire 4.0 or 5.0 Mechanica, or Creo Simulate experience

Audience

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

<table>
<thead>
<tr>
<th>Module</th>
<th>1</th>
<th>Introduction to Creo Simulate</th>
</tr>
</thead>
<tbody>
<tr>
<td>Module</td>
<td>2</td>
<td>Model Preparation and Analysis</td>
</tr>
<tr>
<td>Module</td>
<td>3</td>
<td>Results Evaluation</td>
</tr>
<tr>
<td>Module</td>
<td>4</td>
<td>Loads and Constraints</td>
</tr>
<tr>
<td>Module</td>
<td>5</td>
<td>Interfaces, Assemblies, and Measures</td>
</tr>
<tr>
<td>Module</td>
<td>6</td>
<td>Modeling and Meshing</td>
</tr>
<tr>
<td>Module</td>
<td>7</td>
<td>Idealizations</td>
</tr>
<tr>
<td>Module</td>
<td>8</td>
<td>Advanced Analysis</td>
</tr>
<tr>
<td>Module</td>
<td>9</td>
<td>Thermal Analysis</td>
</tr>
</tbody>
</table>
Update to Creo Schematics 2.0 from Routed Systems Designer 9.0

Overview

<table>
<thead>
<tr>
<th>Course Code</th>
<th>WBT-3929-0</th>
</tr>
</thead>
<tbody>
<tr>
<td>Course Length</td>
<td>4 Hours</td>
</tr>
</tbody>
</table>

In this course, you will learn about the enhancements in Creo Schematics 2.0. First, you will explore and customize the Schematic Ribbon interface. Then you will become familiar with managing design sheets, using drag handles, and creating custom cable decorations. Next, utilizing the parameter pane, managing dataset display, and assigning parameters using CSV files will be examined. Finally, you will learn how to share design data such as design properties and design sheets.

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

Course Objectives

- Understand interface enhancements
- Understand data enhancements

Prerequisites

- None

Audience

- This course is intended for engineers involved in the schematic 2-D layout of either electrical diagrams or piping diagrams. People in related roles will also benefit from taking this course.
## Table of Contents

<table>
<thead>
<tr>
<th>Module</th>
<th>1</th>
<th>Interface Enhancements</th>
</tr>
</thead>
<tbody>
<tr>
<td>Module</td>
<td>2</td>
<td>Data Enhancements</td>
</tr>
</tbody>
</table>
Advanced Turning and Multi-Task Machining using Creo Parametric 2.0

Overview

<table>
<thead>
<tr>
<th>Course Code</th>
<th>WBT-3930-0</th>
</tr>
</thead>
<tbody>
<tr>
<td>Course Length</td>
<td>16 Hours</td>
</tr>
</tbody>
</table>

In this course, you will learn how to machine components using mill-turn machines, twin-turret machines, and in-line twin spindle lathes. You will also learn how to create turning tool paths and milling tool paths for each machine type, where appropriate. In addition, you will understand how to post-process mill-turn toolpaths and how to use the NC-Check and Vericut material removal simulation tools. Finally, you will learn how to view machine tool simulations for 3-axis milling, 2-axis turning, and 5-axis mill-turn machines.

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

Course Objectives

- Learn the basic mill-turn process
- Create mill-turn work centers
- Create various milling and drilling toolpaths using live tooling
- Learn the basic twin turret turning process
- Create twin turret turning work centers
- Create toolpaths for twin turret lathes
- Learn the basic in-line twin spindle turning process
- Create in-line twin spindle lathe work centers
- Create toolpaths using in-line spindles
- Configure and use multi-tip tools
- Understand mill-turn post processing
- Learn how to use material removal simulation
- Learn how to use machine tool simulation
Prerequisites

- Milling using Creo Parametric 2.0 or Turning using Creo Parametric 2.0

Audience

- This course is intended for manufacturing engineers and NC machinists. People in related roles will also benefit from taking this course.
# Table of Contents

<table>
<thead>
<tr>
<th>Module</th>
<th>1</th>
<th>Mill-Turn Machining</th>
</tr>
</thead>
<tbody>
<tr>
<td>Module</td>
<td>2</td>
<td>Twin Turret Turning</td>
</tr>
<tr>
<td>Module</td>
<td>3</td>
<td>In-Line Twin Spindle Turning</td>
</tr>
<tr>
<td>Module</td>
<td>4</td>
<td>Configuring and Using Multi-Tip Tools</td>
</tr>
<tr>
<td>Module</td>
<td>5</td>
<td>Mill-Turn Post Processing</td>
</tr>
<tr>
<td>Module</td>
<td>6</td>
<td>Material Removal Simulation</td>
</tr>
<tr>
<td>Module</td>
<td>7</td>
<td>Machine Tool Simulation</td>
</tr>
</tbody>
</table>
Freestyle Surface Design using Creo Parametric 2.0

Overview

<table>
<thead>
<tr>
<th>Course Code</th>
<th>WBT-3970-0</th>
</tr>
</thead>
<tbody>
<tr>
<td>Course Length</td>
<td>2 Hours</td>
</tr>
</tbody>
</table>

In this course, you will learn how to use the Freestyle environment to create a primitive shape, such as a sphere, and manipulate the polygonal control mesh to create complex-shaped freeform surface models in Creo Parametric.

In Creo Parametric, you can create freeform surface models using the Style and Freestyle modeling environments. Collectively, the use of these environments is often called Freeform surfacing. The Freestyle modeling environment, the focus of this course, provides commands to create smooth and well-defined B-spline surfaces quickly and easily using a polygonal control mesh.

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

Course Objectives

- Understand the freestyle surface modeling process
- Create freestyle surface models

Prerequisites

- Introduction to Creo Parametric 2.0

Audience

- This course is intended for design engineers and mechanical designers who need to create styled surface geometry. People in related roles will also benefit from taking this course.
# Table of Contents

<table>
<thead>
<tr>
<th>Module</th>
<th>1</th>
<th>Introduction to the Freestyle Surface Modeling Process</th>
</tr>
</thead>
<tbody>
<tr>
<td>Module</td>
<td>2</td>
<td>Creating Freestyle Surface Models</td>
</tr>
</tbody>
</table>
Overview

<table>
<thead>
<tr>
<th>Course Code</th>
<th>WBT-4000-0</th>
</tr>
</thead>
<tbody>
<tr>
<td>Course Length</td>
<td>24 Hours</td>
</tr>
</tbody>
</table>

In this course, you will learn about core modeling skills. This comprehensive, hands-on course is specifically designed for existing SolidWorks users who want to become proficient with Creo Parametric as quickly as possible. Topics include understanding the interface and basic Creo Parametric concepts, selecting and editing, sketching tools, and basic feature, part, and assembly creation. The course also includes a comprehensive design project that enables you to practice your new skills by creating realistic parts. After completing the course, you will be well prepared to work effectively on product design projects using 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

- Learn the basic Creo Parametric modeling process
- 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 extrudes, revolves, and profile ribs
- Utilize internal sketches
- Create sweeps and blends
- Create holes, shells, and drafts
- Create rounds and chamfers
- Assemble with constraints
- Lay out drawings and create views
- Create drawing annotations
- Investigate parent/child relationships
- Resolve failures and seek help
- Complete a comprehensive design project
Prerequisites

- Knowledge of SolidWorks, including an understanding of solid modeling, feature-based, parametric, and associative concepts

Audience

- This course is intended for product designers, drafters, and industrial/conceptual designers. People in related roles can also benefit from taking this course.
# Table of Contents

<table>
<thead>
<tr>
<th>Module</th>
<th>1</th>
<th>Introduction to the Creo Parametric Basic Modeling Process</th>
</tr>
</thead>
<tbody>
<tr>
<td>Module</td>
<td>2</td>
<td>Using the Creo Parametric Interface</td>
</tr>
<tr>
<td>Module</td>
<td>3</td>
<td>Editing Geometry, Features, and Models</td>
</tr>
<tr>
<td>Module</td>
<td>4</td>
<td>Sketcher Geometry and Tools</td>
</tr>
<tr>
<td>Module</td>
<td>5</td>
<td>Creating Extrudes, Revolves, and Ribs</td>
</tr>
<tr>
<td>Module</td>
<td>6</td>
<td>Creating Sweeps and Blends</td>
</tr>
<tr>
<td>Module</td>
<td>7</td>
<td>Creating Holes, Shells, and Draft</td>
</tr>
<tr>
<td>Module</td>
<td>8</td>
<td>Creating Rounds and Chamfers</td>
</tr>
<tr>
<td>Module</td>
<td>9</td>
<td>Project I</td>
</tr>
<tr>
<td>Module</td>
<td>10</td>
<td>Assembling with Constraints</td>
</tr>
<tr>
<td>Module</td>
<td>11</td>
<td>Introduction to Drawings</td>
</tr>
<tr>
<td>Module</td>
<td>12</td>
<td>Resolving Failures and Seeking Help</td>
</tr>
</tbody>
</table>