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Introduction to Creo Parametric 5.0

Overview

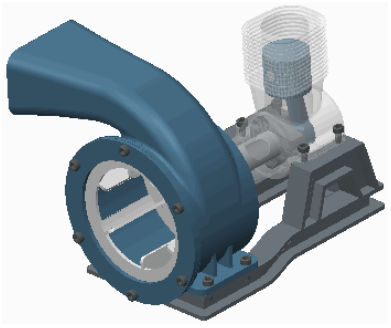
Course Code	TRN-5301-T
Course Length	40 Hours

In this course, you will learn core modeling skills and quickly become proficient with Creo Parametric 5.0. 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 5.0.

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

This course has been developed using Creo Parametric 5.0 F000.

Course Objectives



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

- Explode assemblies
- Lay out drawings and create views
- Create drawing annotations
- Use layers
- Investigate parent/child relationships
- Capture and manage design intent
- Resolve failures and seek help
- Comprehensive two part Design Project

Prerequisites

- None

Audience

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

Agenda

Day 1

Module	1	Introduction to the Creo Parametric Basic Modeling Process
Module	2	Understanding Creo Parametric Concepts
Module	3	Using the Creo Parametric Interface
Module	4	Selecting Geometry, Features, and Models
Module	5	Editing Geometry, Features, and Models
Module	6	Creating Sketcher Geometry

Day 2

Module	7	Using Sketcher Tools
Module	8	Creating Sketches for Features
Module	9	Creating Datum Features: Planes and Axes
Module	10	Creating Extrudes, Revolves, and Ribs
Module	11	Sketcher Workflow
Module	12	Creating Sweeps and Blends

Day 3

Module	13	Creating Holes, Shells, and Draft
Module	14	Creating Rounds and Chamfers
Module	15	Project I
Module	16	Group, Copy, and Mirror Tools
Module	17	Creating Patterns
Module	18	Measuring and Inspecting Models

Day 4

Module	19	Assembling with Constraints
Module	20	Assembling with Connections
Module	21	Exploding Assemblies
Module	22	Drawing Layout and Views
Module	23	Creating Drawing Annotations
Module	24	Using Layers

Day 5

Course Content

Module 1. Introduction to the Creo Parametric Basic Modeling Process

i. Creo Parametric Basic Modeling Process

Module 2. Understanding Creo Parametric Concepts

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

Knowledge Check Questions

Module 3. Using the Creo Parametric Interface

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

Knowledge Check Questions

Module 4. Selecting Geometry, Features, and Models

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

Knowledge Check Questions

Module 5. Editing Geometry, Features, and Models

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

Knowledge Check Questions

Module 6. Creating Sketcher Geometry

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

Knowledge Check Questions

Module 7. Using Sketcher Tools

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

Knowledge Check Questions

Module 8. Creating Sketches for Features

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

Knowledge Check Questions

Module 9. Creating Datum Features: Planes and Axes

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

Knowledge Check Questions

Module 10. Creating Extrudes, Revolves, and Ribs

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

Knowledge Check Questions

Module 11. Sketcher Workflow

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

Knowledge Check Questions

Module 12. Creating Sweeps and Blends

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

Knowledge Check Questions

Module 13. Creating Holes, Shells, and Draft

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

Knowledge Check Questions

Module 14. Creating Rounds and Chamfers

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

Knowledge Check Questions

Module 15. Project I

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

Module 16. Group, Copy, and Mirror Tools

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

Knowledge Check Questions

Module 17. Creating Patterns

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

Knowledge Check Questions

Module 18. Measuring and Inspecting Models

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

vi. Using the Measure Summary Tool
vii. Creating Planar Part Cross-Sections
viii. Measuring Global Interference

Knowledge Check Questions

Module 19. Assembling with Constraints

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

Knowledge Check Questions

Module 20. Assembling with Connections

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

Knowledge Check Questions

Module 21. Exploding Assemblies

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

Knowledge Check Questions

Module 22. Drawing Layout and Views

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

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

Knowledge Check Questions

Module 23. Creating Drawing Annotations

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

Knowledge Check Questions

Module 24. Using Layers

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

Knowledge Check Questions

Module 25. Investigating Parent/Child Relationships

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

Knowledge Check Questions

Module 26. Capturing and Managing Design Intent

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

Knowledge Check Questions

Module 27. Resolving Failures and Seeking Help

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

Knowledge Check Questions

Module 28. Project II

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

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Advanced Modeling using Creo Parametric 5.0

Overview

Course Code	TRN-5302-T
Course Length	24 Hours

The Advanced Modeling Using Creo Parametric 5.0 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 5.0.

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

This course has been developed using Creo Parametric 5.0.

Course Objectives

- Learn advanced selection techniques
- Create advanced datum features
- Use advanced sketching techniques
- Create advanced holes
- Create advanced drafts and ribs
- Create advanced shells
- Create advanced rounds and chamfers
- Use relations and parameters
- Create advanced blends
- Create sweeps with variable sections
- Create helical sweeps and volume helical sweeps (3-D 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 5.0

Audience

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

Agenda

Day 1

Module	1	Advanced Selection
Module	2	Advanced Datum Features
Module	3	Advanced Sketching
Module	4	Advanced Hole Creation
Module	5	Advanced Drafts and Ribs
Module	6	Advanced Shells
Module	7	Advanced Rounds and Chamfers

Day 2

Module	8	Relations and Parameters
Module	9	Advanced Blends
Module	10	Sweeps with Variable Sections
Module	11	Helical Sweeps and Volume Helical Sweep
Module	12	Swept Blends and Advanced Bends

Day 3

Module	13	Advanced Layers
Module	14	Advanced Reference Management
Module	15	Family Tables
Module	16	Reusing Features
Module	17	Advanced Copy
Module	18	Advanced Patterns

Course Content

Module 1. Advanced Selection

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

Knowledge Check Questions

Module 2. Advanced Datum Features

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

Knowledge Check Questions

Module 3. Advanced Sketching

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

Knowledge Check Questions

Module 4. Advanced Hole Creation

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

Knowledge Check Questions

Module 5. Advanced Drafts and Ribs

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

Knowledge Check Questions

Module 6. Advanced Shells

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

Knowledge Check Questions

Module 7. Advanced Rounds and Chamfers

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

xii. Analyzing Advanced Chamfer Dimensioning Schemes
xiii. Analyzing Chamfer Creation Methods
xiv. Creating Corner Chamfers
xv. Creating Chamfers by Reference
xvi. Analyzing Chamfer References and Pieces
xvii. Using Intent Edges for Chamfers
xviii. Using Chamfer Transitions

Knowledge Check Questions

Module 8. Relations and Parameters

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

Knowledge Check Questions

Module 9. Advanced Blends

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

Knowledge Check Questions

Module 10. Sweeps with Variable Sections

i. Understanding Sweeps with Variable Sections Theory
ii. Creating Sweeps Using a Constant Section
iii. Creating Sweeps Normal to Trajectory
iv. Creating Sweeps Using Constant Normal Direction
v. Creating Sweeps with Variable Sections Normal to Projection
vi. Analyzing Horizontal and Vertical Control in Sweeps
vii. Creating Sweeps with Variable Sections Utilizing Multiple Trajectories
viii. Creating Sweeps with Variable Sections Using Tangent Trajectories

ix. Analyzing Sweeps with Variable Sections Trajectory Options and Rules
x. Using Trajpar with Solid Features
xi. Using Trajpar and Datum Graphs with Solid Features

Knowledge Check Questions

Module 11. Helical Sweeps and Volume Helical Sweep

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
vi. Using the Volume Helical Sweep Tool

Knowledge Check Questions

Module 12. Swept Blends and Advanced Bends

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

Knowledge Check Questions

Module 13. Advanced Layers

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

Knowledge Check Questions

Module 14. Advanced Reference Management

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

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Advanced Assembly Design Using Creo Parametric 5.0

Overview

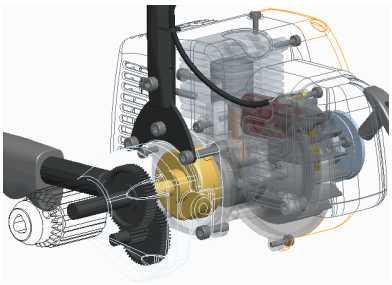
Course Code	TRN-5303-T
Course Length	24 Hours

In this course, you will learn how to use Creo Parametric 5.0 to create and manage complex assemblies. You will discover how to use advanced assembly tools that enable you to add and maintain designs, increase your efficiency, and increase system performance when working with large assemblies. In addition, you will learn the basics of using and creating predefined assembly structures and skeletons, which are both valuable tools typically used in a top-down design process. The course also includes an assembly design project that enables you to practice your new skills by performing various design tasks in an assembly model.

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

This course has been developed using Creo Parametric 5.0.0.0

Course Objectives



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

- Understand advanced simplified rep functionality
- Create and use assembly structure and skeletons
- Utilize design exploration

Prerequisites

- Introduction to Creo Parametric 5.0
- Update to Creo Parametric 4.0

Audience

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

Agenda

Day 1

Module	1	Advanced Component Selection
Module	2	Using Advanced Assembly Constraints
Module	3	Creating and Using Component Interfaces
Module	4	Utilizing Intelligent Fasteners
Module	5	Creating and Using Flexible Components
Module	6	Restructuring and Mirroring Assemblies

Day 2

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

Day 3

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

Course Content

Module 1. Advanced Component Selection

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

Knowledge Check Questions

Module 2. Using Advanced Assembly Constraints

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

Knowledge Check Questions

Module 3. Creating and Using Component Interfaces

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

Knowledge Check Questions

Module 4. Utilizing Intelligent Fasteners

i. Understanding the Intelligent Fastener Extension
ii. Assembling Intelligent Fasteners
iii. Manipulating Intelligent Fasteners
iv. Assembling Intelligent Fasteners Using Advanced Options
v. Manipulating Intelligent Fasteners Using Advanced Options
vi. Inserting Heli-Coils

Knowledge Check Questions

Module 5. Creating and Using Flexible Components

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

Knowledge Check Questions

Module 6. Restructuring and Mirroring Assemblies

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

Knowledge Check Questions

Module 7. Using Assembly Features and Shrinkwrap

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

Knowledge Check Questions

Module 8. Replacing Components in an Assembly

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

Knowledge Check Questions

Module 9. Understanding the Basics of Simplified Reps

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

Knowledge Check Questions

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

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

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

Knowledge Check Questions

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

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

Knowledge Check Questions

Module 12. Understanding Advanced Simplified Rep Functionality

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

Knowledge Check Questions

Module 13. Creating and Using Assembly Structure and Skeletons

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

xiii. Creating Solid Models from a Motion Skeleton

Knowledge Check Questions

Module 14. Utilizing Design Exploration

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

Knowledge Check Questions

Module 15. Project

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

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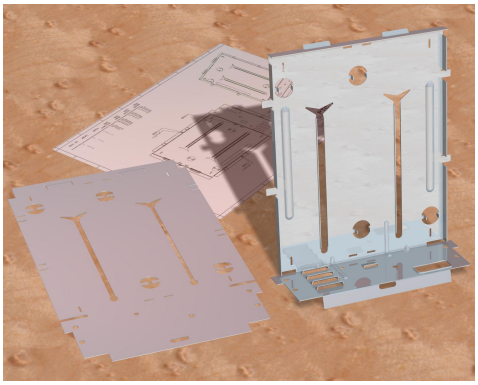
Sheetmetal using Creo Parametric 5.0

Overview

Course Code	TRN-5307-T
Course Length	16 Hours

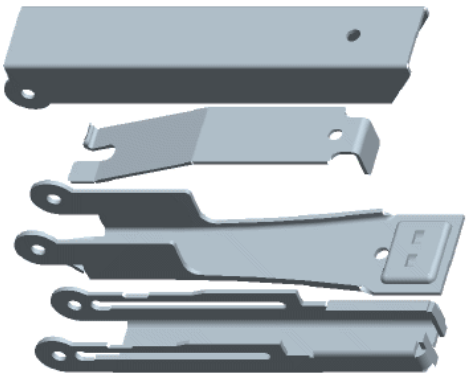


In this course, you will learn how to create sheetmetal parts in Creo Parametric. The course builds upon the basic lessons you learned in Introduction to Creo Parametric 5.0 and serves as the second stage of learning. In this course, you will learn how to design sheetmetal parts and assemblies, including sheetmetal production drawings. All the functions needed to create sheetmetal parts, drawings, and assemblies are covered. Upon completion of this course, you will be able to create sheetmetal design models, create the flat state of the model, and document both in production drawings.

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

This course has been developed using Creo Parametric 5.0 F000.

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 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

Module	1	Introduction to the Creo Parametric Sheetmetal Design Process
Module	2	Sheetmetal Model Fundamentals
Module	3	Creating Primary Sheetmetal Wall Features
Module	4	Creating Secondary Sheetmetal Wall Features

Day 2

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

Course Content

Module 1. Introduction to the Creo Parametric Sheetmetal Design Process

i. Creo Parametric Sheetmetal Design Process

Knowledge Check Questions

Module 2. Sheetmetal Model Fundamentals

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

Knowledge Check Questions

Module 3. Creating Primary Sheetmetal Wall Features

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

Knowledge Check Questions

Module 4. Creating Secondary Sheetmetal Wall Features

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

Knowledge Check Questions

Module 5. Bending and Unbending Sheetmetal Models

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

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

Knowledge Check Questions

Module 6. Sheetmetal Form Features

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

Knowledge Check Questions

Module 7. Modifying Sheetmetal Models

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

Knowledge Check Questions

Module 8. Sheetmetal Setup and Tools

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

Knowledge Check Questions

Module 9. Detailing Sheetmetal Designs

i. Adding Flat and Formed States
ii. Auto Ordinate Dimensions
iii. Bend Line Notes

iv. Bend Order Tables

Knowledge Check Questions

Module 10. Design Project

i. Designing a Stapler

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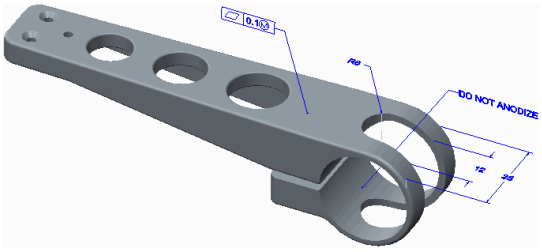
Introduction to Model Based Definition with Creo Parametric 5.0

Overview

Course Code	TRN-5309-T
Course Length	8 Hours



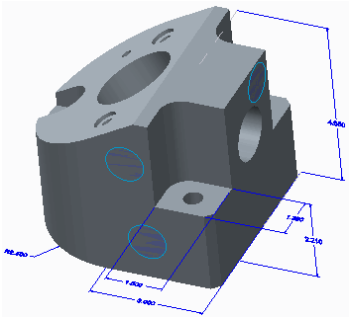
In this course, you will learn how to create an MBD model using a start part. You will discover how annotate mode provides the tools to create an MBD model using the MBD process within Creo Parametric. You will also learn about creating 3-D annotations and how to utilize them in combined states. You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

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

This course has been developed using Creo Parametric 5.0 F000.

Course Objectives

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



Prerequisites

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

Audience

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

Agenda

Day 1

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

Course Content

Module 1. Introduction to Model Based Definition

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

Knowledge Check Questions

Module 2. Preparing Models for Annotation

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

Knowledge Check Questions

Module 3. Creating Annotation Elements

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

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

Knowledge Check Questions

Module 4. Modifying Annotation Elements

i. Modifying Dimension Annotation Display
ii. Manipulating Dimension Annotations
iii. Setting Features with Asymmetric Tolerances to Mid-Spec
iv. Modifying Dimension Annotation Extension Lines
v. Modifying Note Annotations
vi. Controlling Cross-Section Annotation Clipping
vii. Using Security Markings
viii. Modifying Dimension and Note Annotation Arrows
ix. Creating Hyperlinks
x. Identifying and Diagnosing Failing Semantic 3-D Annotations

Knowledge Check Questions

Module 5. Completing Combination States

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

Knowledge Check Questions

Module 6. Publishing for Technical Data Packages

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

viii. Understanding the Creo View User Interface
ix. Orienting the Model in Creo View
x. Viewing Annotations in Creo View
xi. Filtering Entities in the Viewing Area
xii. Creating a STEP AP203 File from Creo Parametric
xiii. Adding Additional Files to a Creo View File

Knowledge Check Questions

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Introduction to PTC Creo Illustrate Professional 5.0

Overview

Course Code	TRN-4559-T
Course Length	8 Hour

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.

This course has been developed using Creo Illustrate 5.0 F000.

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, augmented reality experience authors, and users responsible for parts definition, technical marketing, and service planning. People in related roles will also benefit from taking this course.

Agenda

Day 1

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

Course Content

Module 1. Introduction to Creo Illustrate

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

Module 2. Create an Illustration and Work with Figures

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

Module 3. Exploding Assemblies

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

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

Module 4. Creo Illustrate Animations

i. Figure Animations
ii. Animation Control Tools
iii. Playback Controls
iv. Tracks and Keys
v. Time Line 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

Module 5. 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

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

Module 7. Publishing and Exporting 3-D Illustrations

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

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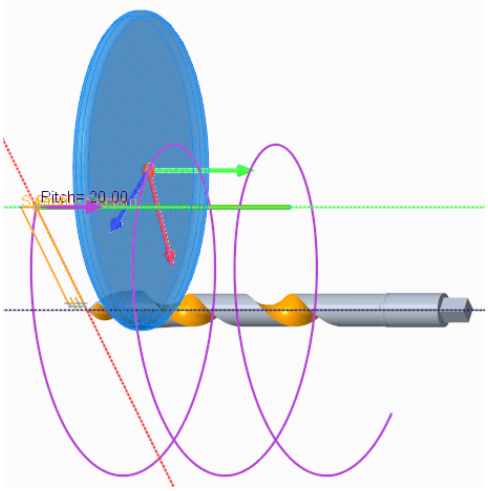
Update to Creo Parametric 5.0 from Creo Parametric 4.0

Overview

Course Code	TRN-5300-T
Course Length	8 Hours



In this course, you will learn how to use the new functionality enhancements in Creo Parametric 5.0, which will enable you to increase your productivity and improve your engineering designs. You will be introduced to new features and enhancements in Part Modeling, Assembly, Drawing, Surfacing, Sheetmetal, and AR design visualization, along with enhancements in Mini Toolbar. You will learn about the new Part Modeling features such as Volume Helical Sweep and Sketch Region selection. You will examine the Part Modeling enhancements to features such as Draft controls for Rounds and Chamfers, Point Pattern workflows, Mirror, and Unite for the Inventor. You will review various Drawing and Detailing enhancements such as improved Undo/Redo support, Arrow Styles, and new configuration options for Cross-hatching Pattern types. You will learn about enhancements to Freestyle Surfacing features such as Align to a non-G2 chain, Align Curvature Connection, Enable Box Creation Mode, Add Edge functionality, and Symmetric Style Curve, along with a new feature for Slice Mesh by Plan. You will be introduced to enhancements of the Sheetmetal design such as Closed sections for the Bend Relief, new Corner Relief types, Sheetmetal Part Conversion, improved Sheetmetal Rounds and Chamfer behavior, and Flattened Representations. You will also be introduced to enhanced Mini Toolbar for the Style, For 3-D and 2-D Annotations, and Sketch Regions.

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

This course has been developed using Creo Parametric 5.0 F000

Course Objectives

- Use the Interface enhancements in Creo Parametric 5.0
- Use the Part Modeling enhancements in Creo Parametric 5.0
- Use the Sketcher enhancements in Creo Parametric 5.0
- Use the Assembly Modeling enhancements in Creo Parametric 5.0
- Use the Drawing enhancements in Creo Parametric 5.0
- Use the Surfacing enhancements in Creo Parametric 5.0
- Use the Sheetmetal enhancements in Creo Parametric 5.0

Prerequisites

- Introduction to Creo Parametric 5.0, or equivalent experience with Creo Parametric 4.0

Audience

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

Agenda

Day 1

Module	1	Interface Enhancements
Module	2	Part Modeling Enhancements
Module	3	Assembly Enhancements
Module	4	Drawing Enhancements
Module	5	Surfacing Enhancements
Module	6	Sheetmetal Enhancements
Module	7	Augmented Reality Design for Visualization

Course Content

Module 1. Interface Enhancements

i. Using the Mini Toolbar
ii. Using Modernized Appearance and Graphics
iii. Using Model Tree Default Filter
iv. Searching in the Model Tree
v. Using Show and Hide Commands
vi. Designing in Perspective View

Knowledge Check Questions

Module 2. Part Modeling Enhancements

i. Using the Volume Helical Sweep Tool
ii. Handling Rounds and Chamfers in a Draft
iii. Creating Point Pattern
iv. Creating a Mirror Feature
v. Using Sketch Regions

Knowledge Check Questions

Module 3. Assembly Enhancements

i. Using Collapsible List for Easy Fastener Selection
ii. Inserting Heli-Coils
iii. Using Unite for the Inventor

Knowledge Check Questions

Module 4. Drawing Enhancements

i. Using Radial Dimensions
ii. Using Undo and Redo Options
iii. Additional Report Parameters in Detailed Drawings
iv. Creating Cross-Hatching
v. Using ISO-Compliant Leader Notes in Detailed Drawings
vi. Understanding Duplicate Draft Datum Feature Symbols Check
vii. Using Rotating Notes in Detailed Drawings

Knowledge Check Questions

Module 5. Surfacing Enhancements

i. Using Align Command to Non-G2 Chains in Freestyle
ii. Using the Align Curvature Connection Command
iii. Creating Slice Mesh by Plane
iv. Enabling the Box Creation Mode
v. Understanding Magnet Behavior on Add Edge in Style
vi. Creating Symmetric Curve in Style
vii. Using Flip Loft Normal Connection
viii. Creating Mirror Curves in Style

ix. Understanding Resolve Mode in Style

Knowledge Check Questions

Module 6. Sheetmetal Enhancements

i. Using Closed Section for Bend Relief
ii. Using Corner Relief
iii. Converting Solid Models to Sheetmetal
iv. Understanding Rounds and Chamfer Behavior
v. Creating Flattened Representation

Knowledge Check Questions

Module 7. Augmented Reality Design for Visualization

i. Understanding Augmented Reality Experience for CAD Design
ii. Setting a Spatial Target
iii. Publishing and Viewing Augmented Reality Experience
iv. Sharing an Experience Model

Knowledge Check Questions