Creo Parametric 5.0 for SolidWorks Users

Overview

Course Code  TRN-5326-T
Course Length  24 Hours

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

This course has been developed using Creo Parametric 5.0.1.0.

Course Objectives

• Create a basic Creo Parametric model
• 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

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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. Using the Command Search Settings for SolidWorks
   vi. Analyzing Basic 3-D Orientation
   vii. Creating and Managing View Orientations
   viii. Setting Up New Part Models
   ix. Understanding Creo Parametric Basic Controls
   x. Understanding the Model Tree
   xi. Selecting Items Using Direct Selection
   xii. Selecting Items Using Query Selection
   xiii. Using the Search Tool
   xiv. Using the Geometry Selection Filter
   xv. 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. Importing SolidWorks Data into Creo Parametric
   ix. Using the Shape Selection Workflow
   x. Applying Flexible Move Using the 3D Dragger
   xi. 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
x. 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. Creating Profile Rib Features
ix. 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
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 Default Constraint
vi. Orienting Components
vii. Creating Coincident Constraints Using Geometry
viii. Assembling Using Automatic

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

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