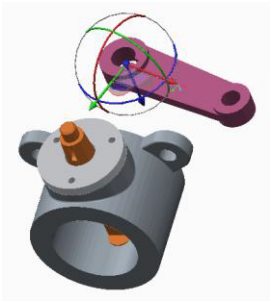
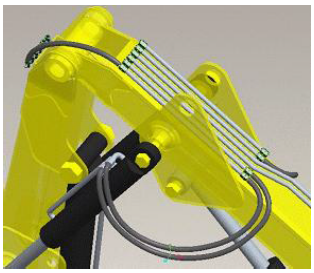
Creo for Piping & Cabling Designers

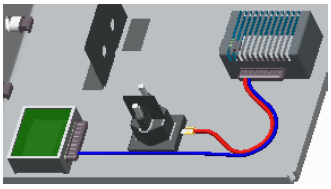
Overview



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 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 and enhancements for dragging components.



In this course, you will learn how to manually create (non-specification driven) mechanical piping designs using Creo Parametric. This includes learning how to configure pipelines, how to route pipelines, and how to insert pipe fittings such as valves and reducers. You will learn how to document piping designs by creating drawings that include BOM tables, pipe bend tables, and engineering information.



In this course, you will learn how to create 3-D electrical harnesses using Creo Parametric. You will learn how to route electrical harnesses 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. After successfully completing the course, you will be able to create 3-D electrical harnesses and associated manufacturing deliverables using Creo Parametric.

Course Objectives

- Introduction & Understanding to Creo Parametric Concepts
- Using Creo Parametric Interface
- Selecting & Editing of Geometry, Features, Models
- Creating Sketcher Geometry & Using Sketcher Tools
- Using Sketches & Datum Features
- Creating Extrudes & Revolves
- Creating Holes, Shells, Draft & Patterns
- Creating Rounds, Chamfers & Using Layers
- Assembling with Constraints
- Exploding, Replacing Components, Cross-Sections in Assemblies
- Introduction to Piping
- Configuring and Routing Pipelines
- Fittings & Solid Pipeline
- Piping Information & Drawings
- Introduction to Cabling
- Setting Up for Cabling
- Routing & Modifying Wires
- Flat Harnesses & Documenting
- Prerequisites
- None

Audience

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

Duration

- 72 Hrs (9 Days)
  - *40 Hrs for ATC's Part (Basic) & Assembly Modeling (Basic) - Creo 2.0*
  - *32 Hrs for ATC's Piping & Cabling - Creo 2.0*

Agenda

ATC's Part (Basic) & Assembly Modeling (Basic) - Creo 2.0

1. Introduction & Understanding to Creo Parametric Concepts

- Creo Parametric Basic Modeling Process
- Understanding Solid Modeling Concepts
- Understanding Feature-Based Concepts
- Understanding Parametric Concepts
- Understanding Associative Concepts
- Understanding Model-Centric Concepts
- Recognizing File Extensions

2. Using Creo Parametric Interface

- Understanding the Main Interface
- Understanding the Folder Browser
- Setting the Working Directory and Opening and Saving Files
- Understanding the Ribbon Interface
- Managing Files in Creo Parametric
- Understanding Datum Display Options
- Analyzing Basic 3-D Orientation
- Understanding the View Manager
- Setting Up New Part Models

3. Selecting & Editing of Geometry, Features, Models

- Understanding Creo Parametric Basic Controls
- Using Drag Handles and Dimension Draggers
- Understanding the Model Tree
- Selecting Items using Direct Selection
- Selecting Items using Query Selection
- Using the Smart Selection Filter
- Utilizing Undo and Redo Operations
- Understanding Regeneration and Auto Regeneration

- Editing Features
- Editing Features using Edit Definition
- Deleting and Suppressing Items

4. Creating Sketcher Geometry & Using Sketcher Tools

- Reviewing Sketcher Theory
- Understanding Design Intent
- Utilizing Constraints
- Sketching Lines
- Sketching Rectangles and Parallelograms
- Sketching Circles
- Sketching Arcs
- Understanding Construction Geometry Theory
- Using Geometry Tools within Sketcher
- Dimensioning Entities within Sketcher
- Modifying Dimensions within Sketcher

5. Using Sketches & Datum Features

- Creating Sketches ('Sketch' Feature)
- Specifying and Manipulating the Sketch Setup
- Utilizing Sketch References
- Using Entity from Edge within Sketcher
- Creating Datum Features Theory
- Creating Datum Axes
- Creating Datum Planes

6. Creating Extrudes & Revolves

- Creating Solid Extrude Features
- Adding Taper to Extrude Features
- Common Dashboard Options: Extrude Depth
- Creating Solid Revolve Features
- Common Dashboard Options: Revolve Angle

7. Creating Holes, Shells, Draft & Patterns

- Common Dashboard Options: Hole Depth
- Creating Coaxial Holes
- Creating Linear Holes
- Creating Radial and Diameter Holes
- Creating Shell Features
- Creating Draft Features
- Creating Basic Split Drafts
- Direction Patterning in the First Direction
- Axis Patterning in the First Direction

- Creating Reference Patterns of Features

8. Creating Rounds, Chamfers & Using Layers

- Creating Rounds Theory
- Creating Rounds by Selecting Edges
- Creating Rounds by Selecting a Surface and Edge
- Creating Rounds by Selecting Two Surfaces
- Creating Full Rounds
- Creating Chamfers by Selecting Edges
- Analyzing Basic Chamfer Dimensioning Schemes
- Understanding Layers
- Utilizing Layers in Part Models
- Creating and Managing Layers

9. Assembling with Constraints

- Understanding Assembly Theory
- Creating New Assembly Models
- Understanding Constraint Theory
- Assembling Components using the Default Constraint
- Creating Coincident Constraints using Geometry
- Creating Coincident Constraints using Datum Features
- Creating Distance Constraints
- Creating Parallel, Normal, and Angle Constraints
- Assembling using Automatic

10. Exploding, Replacing Components, Cross-Sections in Assemblies

- Creating and Managing Explode States
- Animating Explode States
- Understanding Component Replace
- Replacing Components using Family Table
- Understanding Assembly Cross-Sections
- Creating Assembly Cross-Sections
- Creating Offset Assembly Cross-Sections
- Creating Display Styles

ATC's Piping & Cabling - Creo 2.0

11. Introduction to Piping

- Understanding Piping Design Methods
- Manual Piping Development Process
- Understanding Piping Terminology

12. Configuring and Routing Pipelines

- Understanding Pipeline Routing
- Configuring Non-Specification-Driven Pipelines
- Routing Pipelines
- Routing Flexible Hoses

13. Fittings & Solid Pipeline

- Understanding Fittings
- Creating Fittings
- Inserting Fittings
- Creating Solid Pipes
- Extracting Models

14. Piping Information & Drawings

- Using Piping Reporting Tools
- Creating Piping Drawings
- Displaying Piping Report Parameters

15. Introduction to Cabling

- Step 1: Assembly and Cabling Setup
- Step 2: Routing Wires and Cables
- Step 3: Flattening the Harness
- Step 4: Creating the Harness Drawing

16. Setting Up for Cabling

- Creating and Configuring Connectors
- Assembling Connectors
- Understanding the Cabling Interface
- Creating a Harness Part
- Creating a Wire Color Appearance File
- Manually Designating Connector and Entry Ports
- Creating Wire Spools

17. Routing & Modifying Wires

- Routing Wires using Simple Route
- Inserting and Editing Wire Locations
- Rerouting Wires
- Deleting Wires and Segments
- Editing Location Properties
- Modifying Wire Packing
- Creating Bundles

18. Flat Harnesses & Documenting

- Creating a Flat Harness Model
- Using Auto Fan
- Assembling Harness Components

- Creating Cabling Assembly Views
- Creating Harness Views
- Placing Spool BOM Tables
- Placing Harness From and To Tables