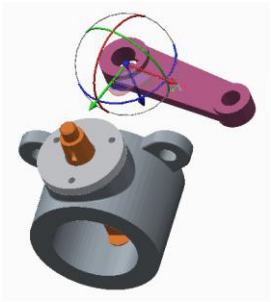
Creo for Design Engineers

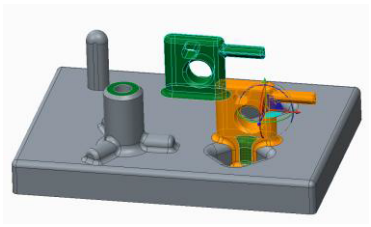
Overview

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.



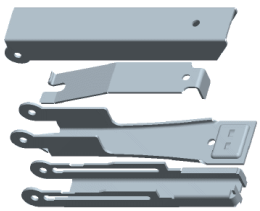
The Advanced Modeling using Creo Parametric 2.0 teaches you how to use advanced part modeling techniques to improve your product designs. In this course, you will learn how to create and modify design models using advanced sketching techniques and feature creation tools. You will also learn how to reuse 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.

In this, 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. Each stage of the process is described in detail and supported by step-by-step exercises.

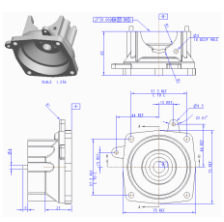


In this, you will learn how to use Creo Parametric 2.0 to create and manage complex assemblies. You will learn how to use advanced assembly tools that enable you to add and maintain designs, increase your efficiency, and increase system performance when working with large assemblies. In addition, you 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.

Sheetmetal Design using Creo Parametric 2.0 is a comprehensive 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. All the functions needed to create sheetmetal parts are covered. Upon completion of this course, you will be able to create sheetmetal design models, create the flat state of the model.



Detailed drawings teaches you how to quickly create detailed drawings using information captured within 3-D design models. In this course, you will 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.



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
- Advanced Selection, Creating Sweeps and Blends
- Sweeps with Variable Sections
- Helical Sweeps & Swept Blends
- Relations, Parameters & Family Tables
- Groups, Copy, Mirror & UDF's
- Measuring, Inspecting Models & Seeking Help
- Capturing, Managing Design Intent & Resolving Failures
- Introduction to Flexible Modeling
- Editing, Transformations & Recognition in Flexible Modeling
- Component Interfaces, Flexible Components, Restructuring, Simplified Reps
- Creating and Using Assembly Structure and Skeletons
- Sheetmetal Design Process & Fundamentals
- Creating Primary & Secondary Sheetmetal Wall Features
- Bending, Unbending & Modifying Sheetmetal Models
- Introduction, Creating New Drawings & Drawing Views
- Adding Model Details &Tolerance Information to Drawings
- Adding Notes, Symbols, Tables, Balloons & Layers in Drawings

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

- 108 Hrs (13.5 Days)
  - *40 Hrs for ATC's Part (Basic) & Assembly Modeling (Basic) - Creo 2.0*
  - *36 Hrs for ATC's Part (Adv) & Flexible Modeling - Creo 2.0*
  - *32 Hrs for ATC's Assembly (Adv), Sheetmetal Modeling & Detailing - 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 Part (Adv) & Flexible Modeling - Creo 2.0

11. Advanced Selection, Creating Sweeps and Blends

- Advanced Chain Selection
- Advanced Surface Selection
- Creating Sweeps with Open Trajectories
- Creating Sweeps with Closed Trajectories
- Creating Blends by Sketching Sections
- Creating Rotational Blends by Selecting Sections

12. Sweeps with Variable Sections

- Understanding Sweeps with Variable Sections Theory
- Creating Sweeps Normal to Trajectory
- Creating Sweeps using Constant Normal Direction
- Creating Sweeps with Variable Sections Normal to Projection
- Creating Sweeps with Variable Sections Utilizing Multiple Trajectories

13. Helical Sweeps & Swept Blends

- Understanding Helical Sweeps Theory
- Creating Helical Sweeps for Springs
- Understanding Swept Blend Theory
- Creating Swept Blends by Selecting Sections
- Creating Swept Blends by Sketching Sections

14. Relations, Parameters & Family Tables

- Understanding Relation Theory
- Understanding Relation Types
- Creating Parameters
- Creating Relations
- Understanding Family Table Theory
- Creating a Family Table

15. Groups, Copy, Mirror & UDF's

- Creating Local Groups
- Copying and Pasting Features
- Moving and Rotating Copied Features
- Mirroring Selected Features
- Creating UDF's
- Placing UDF's

16. Measuring, Inspecting Models & Seeking Help

- Viewing and Editing Model Properties
- Investigating Model Units
- Analyzing Mass Properties
- Creating Planar Part Cross-Sections
- Using Creo Parametric Help

17. Capturing, Managing Design Intent & Resolving Failures

- Handling Children of Deleted and Suppressed Items
- Reordering Features
- Inserting Features
- Redefining Features and Sketches
- Understanding and Identifying Failures
- Analyzing Geometry Failures
- Analyzing Open Section Failures
- Analyzing Missing Part Reference Failures

18. Introduction to Flexible Modeling

- Understanding Flexible Modeling
- Flexible Modeling Process
- Using the Selection Filter
- Applying Shape Selection

- Applying Boss Selections

19. Editing, Transformations & Recognition in Flexible Modeling

- Applying Flexible Move using Dragger
- Using Flexible Mirror
- Using the Edit Round Feature
- Working with Pattern Recognition
- Using the Flexible Attach Feature

ATC's Assembly (Adv), Sheetmetal Modeling & Detailing- Creo 2.0

20. Component Interfaces, Flexible Components, Restructuring, Simplified Reps

- Understanding Component Interfaces
- Using a Placing Component Interface
- Using a Receiving Component Interface
- Adding Flexibility to a Component
- Placing Flexible Components in an Assembly
- Restructuring and Reordering Assembly Components
- Understanding Standard Simplified Reps
- Excluding Components using Simplified Reps

21. Creating and Using Assembly Structure and Skeletons

- Understanding Skeletons
- Creating Assembly Structure
- Creating Skeletons for Space Claims
- Creating Skeletons for Placement References
- Copying a Model to a Skeleton
- Creating Multiple Skeletons
- Sharing Skeleton Geometry
- Creating and Placing Models using Skeleton References

22. Sheetmetal Design Process & Fundamentals

- Creo Parametric Sheetmetal Design Process
- Sheetmetal Model Fundamentals
- Understanding Developed Length
- Creating a New Sheetmetal Model in Part Mode

23. Creating Primary & Secondary Sheetmetal Wall Features

- Understanding Sheetmetal Wall Features
- Creating Planar Walls
- Extruded Sheetmetal Wall Features
- Revolved Sheetmetal Wall Features
- Understanding Secondary Walls

- Creating Secondary Flat Walls
- Using Flange Walls
- Using Extruded Walls
- Understanding Relief

24. Bending, Unbending & Modifying Sheetmetal Models

- Creating Bend Features
- Adding Transition to Bends
- Creating Unbend Features
- Creating Bend Back Features
- Creating Flat States
- Sheetmetal Cuts
- Die Form Features
- Punch Form Features
- Creating Rip Features

25. Introduction, Creating New Drawings & Drawing Views

- Understanding Drawing Concepts
- Exploring Drawing Ribbon Commands
- Creating Drawings Using Formats and Sheets
- Configuring Drawing Models
- Adding General Views
- Adding Projection Views
- Editing Drawing Views
- Editing Visible View Area
- Adding Detailed Views
- Adding 2-D Cross-Section Views
- Adding Assembly Exploded Views

26. Adding Model Details &Tolerance Information to Drawings

- Understanding Annotations in Drawings
- Showing, Erasing, and Deleting Annotations
- Adjusting Dimensions and Detail Items
- Changing Dimension Display
- Understanding Dimensional Tolerances
- Configuring Dimensional Tolerances
- Understanding Geometric Tolerances
- Setting Up Geometric Tolerance References
- Applying Geometric Tolerances

27. Adding Notes, Symbols, Tables, Balloons & Layers in Drawings

- Adding and Editing Notes
- Using Surface Finish Symbols

- Inserting Tables
- Creating Report Tables
- Creating BOM Balloons
- Understanding Layers in Drawings
- Using Layers in Drawings
- Configuring the Drawing Environment