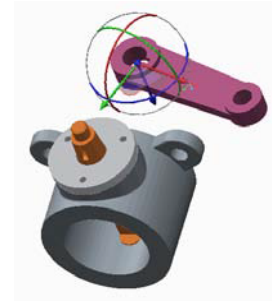


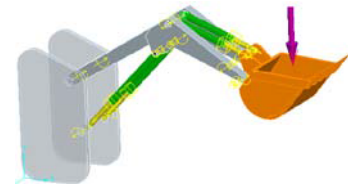
Creo for Analyst

Overview

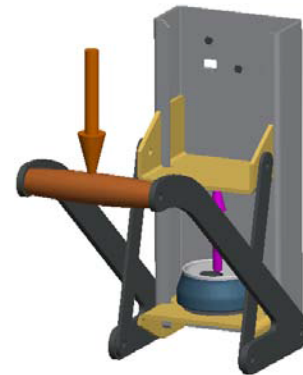
In this course, you will learn how to utilize the core functionality enhancements in Creo Parametric 2.0. First, you will become familiar with using and customizing the new ribbon interface in Creo Parametric. The new measure and sectioning interfaces will also be examined. Next, you will become familiar 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.



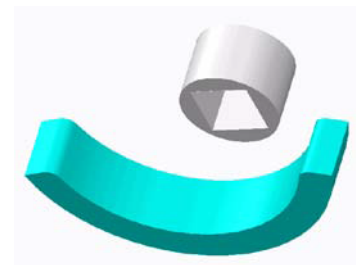
In this you will learn about creating mechanism connections, configuring the mechanism model, creating a kinematic analysis, and evaluating results. Mechanism Design using Creo Parametric is designed for experienced users who want to add motion to their models by creating mechanism connections and servo 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.



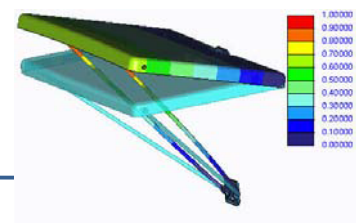
In this, you will focus on learning advanced modeling and analysis skills. Topics will include developing the 3-D model, analyzing the mechanism model, and evaluating results. This course is designed for experienced users who want to add motion to their products and analyze dynamic reactions of moving 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).



In this, 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 aid you in determining 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 in order to meet your design specifications. This course is designed for experienced users who want to add additional features that enable you to meet or exceed the design specifications of your products.



This course is designed for new users who want to test, validate, and optimize product designs with the Creo Simulate module. Simulate enables you to simulate structural and thermal loads on product designs. In this course, 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.

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 Mechanism Design
- Introduction to Mechanism Simulation
- Introduction to Behavioral Modeling
- Introduction to Creo Simulate & Theoretical Foundations
- Model Preparation & Analysis Definition in Creo Simulate
- Loads, Constraints and Meshing in Creo Simulate
- More Analysis Types in Creo Simulate

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

- 84 Hrs (10.5 Days)
 - 40 Hrs for ATC's Part (Basic) & Assembly Modeling (Basic) - Creo 2.0
 - 44 Hrs for ATC's Analysis in Creo - 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 Analysis in Creo - Creo 2.0

11. Introduction to Mechanism Design

- Introduction to Mechanism Design
- Understanding the Mechanism Design Process
- Creating the Model
- Verifying the Mechanism
- Adding Servo Motors
- Preparing for Analysis of a Mechanism
- Analyzing the Mechanism
- Evaluating Analysis Results

12. Introduction to Mechanism Simulation

- Introduction to Mechanism Simulation
- Understanding the Mechanism Simulation Process
- Creating the Model
- Verifying the Mechanism

- Adding Dynamic Entities
- Preparing for Analysis of a Mechanism
- Analyzing the Mechanism
- Evaluating Analysis Results
- Graphing Measure Results

13. Introduction to Behavioral Modeling

- Behavioral Modeling Process
- Identifying BMX Analysis Types
- Identifying the Differences Between Creo Parametric Analyses
- Performing Sensitivity Analysis

14. Introduction to Creo Simulate & Theoretical Foundations

- Simulate Analysis Functionality
- Simulate Model Functionality
- The Finite Element Method
- The Typical Simulation Process
- The h and p Versions of Finite Elements
- The p-Method
- Structural Mechanics – Stress Definitions and Hooke's Law

15. Model Preparation & Analysis Definition in Creo Simulate

- Managing Units
- Defining Linear Elastic Materials
- Convergence Methods
- Fundamentals of a Linear Static Analysis
- Defining a Linear Static Analysis
- Selecting a Convergence Method

16. Loads, Constraints and Meshing in Creo Simulate

- Defining Forces, Moments, and Pressure
- Defining Constraints
- Using AutoGEM Settings

17. More Analysis Types in Creo Simulate

- Using Modal Analysis
- Understanding Symmetry
- Using Shell Pairs for Midsurface Models
- Understanding Dynamic Analysis
- Understanding Linear Buckling Analysis