Creo Parametric

Course Objective

Upon successful completion of this course, participants will gain a comprehensive understanding of Creo Parametric, a powerful parametric 3D computer-aided design (CAD) software. The course aims to equip students with the foundational skills necessary to effectively utilize Creo Parametric for modeling, assembly design, and engineering documentation

Course Outcome

By the end of this course, participants will have acquired a solid foundation in Creo Parametric, empowering them to confidently model, analyze, and document engineering designs. The skills gained in this course will prepare students for various applications in the fields of product design, engineering, and manufacturing.

Course Outline

The course comprises 40-hours of theory and labs and is divided into 12 different modules.

Chapter 1. Introduction to Creo Parametric

- Installing Creo Parametric
- Getting Started with Creo Parametric
- Starting a New File
- Identifying Creo Parametric Files
- Setting the Working Directory
- Erasing Objects of the Current Session
- Saving an Object
- Saving a Copy of an Object
- Opening an Object
- Deleting Older Versions of a Design
- Changing Background Color
- Customizing the Ribbon
- Customizing the Quick Access Toolbar
- Closing a File

Chapter 2. Drawing Sketches and Applying Dimensions

- Invoking the Part Mode
- Specifying Units
- Invoking the Sketching Environment
- Working with Selection of Planes
- Specifying Grids and Snap Settings
- Drawing Sketch Entities
- Creating Datum Geometries
- Working with Construction Mode
- Applying Constraints
- Working with Weak Dimensions
- Applying Dimensions
- Editing Dimensions

Chapter 3. Editing and Modifying Sketches

- Trimming Sketch Entities
- Creating Corners by Trimming/Extending Entities
- Dividing Entities
- Moving, Rotating, and Scaling Entities
- Mirroring Entities
- Offsetting Entities
- Offsetting Entities Bi-directionally
- Creating a Sketch Fillet
- Creating a Sketch Chamfer
- Inserting Text in Sketching Environment
- Working with inspection tools
- Inserting Pre-defined Shapes of Geometries
- Inserting a Sketch or a Drawing File

Chapter 4. Creating Base Feature of a Solid Model

- Creating an Extrude Feature
- Creating a Revolve Feature
- Navigating a 3D Model in Graphics Window
- Manipulating View Orientation of a Model
- Changing the Display Style of a Model

Chapter 5. Creating Datum Geometries

- Creating Datum Planes
- Creating a Datum Axis
- Creating a Datum Coordinate System
- Creating a Datum Point

Chapter 6. Advanced Modeling - I

- Using Advanced Options of the Extrude Tool
- Using Advanced Options of the Revolve Tool
- Projecting Edges onto the Sketching Plane
- Editing a Feature
- Editing the Sketch of a Feature
- Measuring Geometries
- Assigning an Appearance
- Editing an Appearance
- Copying and Pasting an Appearance
- Removing Appearances
- Applying a Material
- Calculating Mass Properties

Chapter 7. Advanced Modeling - II

- Creating a Sweep Feature
- Creating a Helical Sweep feature
- Creating a Volume Helical Sweep feature
- Creating a Blend feature
- Creating a Swept Blend feature
- Creating a Rotational Blend feature

Chapter 8. Patterning and Mirroring

- Creating a Dimension Pattern
- Creating a Direction Pattern
- Creating an Axis Pattern
- Creating a Fill Pattern
- Creating a Table Pattern
- Creating a Reference Pattern
- Creating a Curve Pattern
- Creating a Point Pattern
- Creating a Variable Pattern
- Creating a Geometry Pattern
- Deleting a Pattern
- Mirroring a Feature

- Copying and Pasting a Feature

Chapter 9. Advanced Modeling - III

- Creating Simple and Standard Holes
- Creating Cosmetic Threads
- Creating Rounds
- Creating Auto Rounds
- Creating Chamfers
- Creating Rib Features
- Creating Shell Features

Chapter 10. Working with Assemblies - I

- Bottom-up Assembly Approach
- Top-down Assembly Approach
- Invoking the Assembly Mode
- Creating an Assembly by using Bottom-up Approach
- Fixing the First Assembly Component
- Applying Constraints
- Moving/Rotating Individual Components
- Editing Constraints

Chapter 11. Working with Assemblies - II

- Creating an Assembly using the Top-down Approach
- Editing Assembly Components
- Displaying Constraints in Model Tree
- Patterning Assembly Components
- Mirroring a Component of an Assembly
- Creating Assembly Features
- Suppressing or Resuming Components
- Assembling Multiple Copies of a Component
- Checking Interference between Components
- Creating an Exploded View
- Switching Between Exploded and Unexploded Views
- Specifying Settings for Animating Exploded View
- Creating Bill of Material (BOM)

Chapter 12. Working with Drawings

- Invoking Drawing Mode
- Adding a Model for Generating its Views
- Creating a General View
- Creating Projection Views
- Working with Angle of Projection
- Defining the Angle of Projection
- Creating a Detailed View
- Creating an Auxiliary View
- Creating a Revolved View
- Creating a Section View
- Controlling the Visibility of a View
- Creating a 3D Cross-Section View
- Creating a Copy and Align View
- Modifying Properties of a View
- Modifying Hatching of a View
- Moving, Erasing, and Deleting a View
- Creating a New Drawing Template/Format
- Applying Dimensions
- Editing the Text Style
- Adding Tolerances in the Drawing Views
- Adding Notes
- Creating the Bill of Material (BOM)
- Adding Balloons