

Creo Parametric: Basic to Advance with Simulation & Analysis

Course Objective

This course aims to provide participants with comprehensive, industry-oriented proficiency in Creo Parametric for mechanical design, simulation, and engineering documentation. It focuses on developing strong skills in 3D modeling, assemblies, drawings, and advanced surface modeling while preserving design intent. The course also introduces mechanism design, finite element analysis, and fluid flow analysis to enable design validation and performance evaluation. By integrating CAD and CAE workflows, the program prepares learners to apply Creo Parametric effectively in real-world product design, engineering, and manufacturing environments

Course Outcome

Upon completion of this course, learners will be able to create accurate 3D models, assemblies, and engineering drawings using Creo Parametric. They will gain the ability to perform basic mechanism design, surface modeling, structural analysis, and fluid flow analysis for design validation. The acquired skills will enable participants to confidently apply Creo Parametric in practical product design and manufacturing applications

Course Outline

The course comprises 64-hours of theory and labs and is divided into 16 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 . Creo Mechanism design

- Introduction to Creo Mechanism
- Mechanism bodies and connections
- Joint types (pin, slider, cylinder, planar, ball)
- Servo motors and motion drivers
- Position, velocity, and acceleration analysis
- Kinematic motion simulation
- Collision detection and interference checking
- Tracing paths and measuring motion results
- Mechanism playback and result visualization
- Exporting mechanism results

Chapter 13. 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

Chapter 14. Surface Modelling & Boundary conditions

- Overview of surface modelling in Creo
- Solid vs surface vs hybrid modelling approach
- Role of surfaces in complex geometry design

Curves & Reference Creation

- Datum curves and curve features
- Curve through points
- Curve from equation
- Projected curves
- Composite and intersect curves

Surface Creation Techniques

- Extruded surfaces
- Revolved surfaces
- Swept surfaces
- Boundary blend surfaces
- Fill surfaces
- Offset surfaces
- Ruled surfaces
- Variable section surfaces

Advanced Surface Modelling

- Style curves and style surfaces
- Multi-section boundary blends
- Control of surface direction and influence
- Surface alignment and matching
- Surface extension and trimming
- Surface replacement techniques

Surface Continuity & Quality Control

- Positional continuity (G0)
- Tangency continuity (G1)
- Curvature continuity (G2)
- Tangent and curvature boundary control
- Edge alignment and smoothing
- Zebra analysis
- Curvature comb and reflection analysis

Boundary Conditions in Surface Modelling

- Defining boundary edges
- Tangency boundary conditions
- Curvature boundary conditions
- Free vs fixed boundary constraints
- Directional influence at boundaries
- Continuity across adjacent surfaces
- Boundary condition impact on surface quality

Surface Editing & Management

- Quilt creation and management
- Merging and trimming quilts
- Surface offset and thickening
- Copy, mirror, and pattern surfaces
- Parent-child relationship control

Surface to Solid Conversion

- Closing open surface quilts
- Solidify and thicken operations
- Gap detection and closure
- Validation of watertight geometry

Imported Surface Handling

- Import diagnostics
- Gap and overlap detection
- Boundary repair and healing
- Rebuilding poor-quality boundaries
- **Best Practices & Industry Workflow**
- Surface modelling strategy planning
- Boundary-driven design approach
- Common surface and boundary errors
- Design intent preservation
- Industry best practices for Class-A readiness

Chapter 15. Finite Element Analysis

- The Direct Stiffness Method
- Truss Elements in Two-Dimensional Spaces
- 2D Trusses in MS Excel and the Truss Solver
- Creo Simulate Two-Dimensional Truss Analysis
- Three-Dimensional Truss Analysis
- Basic Beam Analysis
- Beam Analysis Tools
- Statically Indeterminate Structures
- Two Dimensional Solid Elements
- Three Dimensional Solid Elements
- Axisymmetric and Thin Shell Elements
- Dynamic Modal Analysis

Chapter 16. Fluid Flow Analysis

- Creo Flow Analysis overview
- Internal flow analysis
- External flow analysis
- Fluid material definition

- Flow domain creation
- Inlet boundary conditions
- Outlet boundary conditions
- Wall boundary conditions
- Initial conditions
- Laminar and turbulent flow selection
- Steady-state flow setup
- Basic transient flow
- Mesh generation
- Local mesh refinement
- Solver settings
- Convergence criteria
- Velocity contours
- Pressure contours
- Streamlines
- Flow vectors
- Mass flow rate calculation
- Pressure drop evaluation
- Flow path visualization
- Design comparison
- Result validation
- Report generation