

FreeCAD: 3D Modeling, Assemblies & Engineering Drawings

Target Audience

This course is designed for mechanical designers, CAD engineers, product designers, manufacturing engineers, drafting professionals, engineering students, and technical professionals who require practical expertise in FreeCAD for parametric mechanical design and engineering documentation. It is suitable for participants with basic CAD or mechanical design knowledge who want to develop industry-ready skills in sketch-based modeling, advanced feature creation, mechanical assemblies, and professional 2D engineering drawings.

Course Outcomes

After completing this course, participants will be able to:

- **Understand** the FreeCAD design environment, workbenches, document management, and parametric modeling workflow.
- **Create** fully constrained 2D sketches using geometric and dimensional constraints.
- **Develop** parametric 3D mechanical components using feature-based solid modeling techniques.
- **Apply** datum geometry to support complex and reference-driven parametric designs.
- **Create** advanced subtractive, additive, swept, lofted, and helical features.
- **Optimize** mechanical models using patterns, mirrors, holes, fillets, chamfers, shells, and draft features.
- **Build** functional multi-component mechanical assemblies using appropriate assembly joints.
- **Generate** professional 2D engineering drawings from 3D models using TechDraw.
- **Apply** dimensions, section views, detail views, hidden-line representations, and drawing templates.
- **Export** engineering documentation and CAD data in commonly used formats such as DXF and SVG.
- **Complete** an end-to-end mechanical design workflow from constrained sketch through final assembly and technical documentation.

Course Objectives

- Provide structured, hands-on training in FreeCAD for professional mechanical design workflows.

- Develop practical skills in 2D sketch creation, geometric constraint application, and parametric design intent.
- Build proficiency in feature-based 3D solid modeling and advanced mechanical feature development.
- Provide hands-on exposure to datum geometry, feature transformations, patterns, and complex modeling operations.
- Develop practical assembly modeling skills using multiple component types and joint mechanisms.
- Enable participants to create manufacturing-oriented mechanical parts with appropriate dress-up and finishing features.
- Teach professional engineering drawing creation using the TechDraw workbench.
- Develop skills in applying dimensions, section views, detail views, templates, and title-block information.
- Reinforce learning through tutorials, lab exercises, and an end-to-end capstone mechanical design project.
- Prepare participants to apply FreeCAD skills to real-world mechanical design and engineering documentation workflows.

Course Outline

The course comprises **40-hours** of theory and labs and is divided into 10 different chapters. Each chapter will be followed by hands-on lab exercises to reinforce learning and gauge understanding of the topics covered.

Table of Contents

Chapter 1: Getting Started with FreeCAD

- Installing and launching FreeCAD
- Understanding the FreeCAD design environment
- Creating and managing design documents
- Working with FreeCAD Workbenches
- Exploring the FreeCAD User Interface
- Understanding the Toolbar and Standard Menu
- Using the Workbench Selector
- Working with Model and Tasks Panels
- Navigating with the Navigation Cube

- Using Shortcut Menus and Contextual Commands
- Customizing Toolbars and Workspace Layouts
- Managing Display and Background Settings
- Saving, Opening, and Managing FreeCAD Documents
- Exporting Models to Other CAD Formats
- Getting Started with FreeCAD: Hands-on Lab Exercise

Chapter 2: Creating and Constraining 2D Sketches

- Creating a New Design Document
- Working with the Sketcher Workbench
- Selecting and Managing Sketch Planes
- Configuring Units, Grids, and Snap Settings
- Creating Lines, Arcs, and Curves
- Creating Circles and Ellipses
- Creating B-Splines
- Creating Polylines and Rectangles
- Creating Polygons and Slots
- Applying Sketch Fillets and Chamfers
- Trimming, Extending, and Splitting Sketch Geometry
- Creating and Managing Construction Geometry
- Applying Geometric Constraints
- Applying Dimensional Constraints
- Editing and Modifying Sketch Dimensions
- Understanding Under-Constrained Sketches
- Creating Fully Constrained Sketches
- Identifying and Resolving Over-Constrained Sketches
- Sketching Tutorials and Hands-on Exercises

Chapter 3: Building Parametric 3D Solid Models

- Creating Base Features from 2D Sketches
- Creating Pad Features

- Creating Revolved Features
- Understanding the 3D View Environment
- Navigating Parametric 3D Models
- Performing Pan, Zoom, and Rotate Operations
- Using Fit All and Fit Selection
- Working with Standard Model Views
- Using the Navigation Cube for Model Orientation
- Understanding Model Display Styles
- Working with Wireframe Views
- Working with Hidden-Line Views
- Working with Shaded Views
- Creating Base Solid Modeling Projects
- Applying Parametric Design Principles to Solid Models
- 3D Modeling Tutorials and Hands-on Exercises

Chapter 4: Creating Datum Geometry for Advanced Modeling

- Understanding Datum Geometry and Reference Features
- Creating Datum Planes
- Creating Planes Relative to Existing Geometry
- Creating Planes Through Points, Vertices, and Edges
- Creating Tangent and Normal Datum Planes
- Creating Datum Lines and Axes
- Creating Lines Through Points and Along Edges
- Creating Normal and Tangent Datum Lines
- Working with Principal Axes
- Creating Datum Points
- Creating Points on Vertices and Edges
- Creating Points Using Curvature and Conical Geometry
- Creating Local Coordinate Systems
- Applying Datum Geometry in Parametric Modeling

- Managing Reference Geometry for Complex Design Intent
- Datum Geometry Tutorials and Hands-on Exercises

Chapter 5: Creating Pocket, Groove, and Subtractive Features

- Creating Pocket Features
- Creating Groove Features
- Projecting External Geometry into Sketches
- Using Section Views During Sketch Creation
- Reviewing Earlier Model States
- Reordering Model Features
- Deleting and Managing Features
- Editing Features and Associated Sketches
- Modifying Feature Parameters
- Managing Model Appearance and Display Properties
- Measuring Parameters Between Geometries
- Maintaining Parametric Design Relationships During Feature Editing
- Managing Feature Dependencies and Model History
- Applying Subtractive Features to Mechanical Components
- Feature Editing Tutorials and Hands-on Exercises

Chapter 6: Advanced Sweeps, Lofts, and Helical Features

- Creating Additive Pipe Features
- Creating Multi-Section Additive Pipes
- Creating Subtractive Pipe Features
- Creating Additive Loft Features
- Creating Subtractive Loft Features
- Creating Additive Helix Features
- Creating Subtractive Helix Features
- Creating Helical Curves
- Defining Profiles and Paths for Advanced Features
- Managing Multi-Section and Swept Geometry

- Combining Advanced Features in Mechanical Models
- Applying Loft and Sweep Operations to Complex Components
- Developing Threaded and Helical Mechanical Geometry
- Advanced Feature Modeling Tutorials
- Hands-on Advanced Modeling Exercises

Chapter 7: Mirroring and Pattern-Based Modeling

- Understanding Feature Transformation
- Mirroring Features in 3D Models
- Creating Linear Patterns
- Creating Polar Patterns
- Creating Multi-Transform Patterns
- Defining Pattern Directions and Parameters
- Applying Patterns to Mechanical Components
- Managing Repeated Features
- Optimizing Repetitive Geometry Through Parametric Transformations
- Maintaining Associativity in Pattern-Based Designs
- Combining Mirror and Pattern Operations
- Pattern-Based Modeling Tutorials
- Hands-on Pattern Modeling Exercises

Chapter 8: Holes and Mechanical Dress-Up Features

- Creating Hole Features
- Configuring Hole Parameters and Specifications
- Creating Fillets
- Creating Chamfers
- Creating Thin-Wall and Shell Models
- Creating Angular Draft Features
- Applying Dress-Up Features to Mechanical Parts
- Managing Feature Order for Dress-Up Operations
- Refining Parametric Models for Manufacturing Intent

- Applying Edge Treatments and Surface Modifications
- Developing Manufacturing-Oriented Part Geometry
- Reviewing Mechanical Feature Dependencies
- Mechanical Feature Modeling Tutorials
- Hands-on Part Design Exercises

Chapter 9: Creating and Managing Mechanical Assemblies

- Understanding Assembly Modeling Concepts
- Creating Multi-Body Assemblies
- Creating Bottom-Up Assemblies
- Working with the Assembly Workbench
- Inserting Components into Assemblies
- Positioning and Connecting Components
- Applying Assembly Joints
- Creating Fixed Joints
- Creating Revolute Joints
- Creating Cylindrical Joints
- Creating Slider Joints
- Creating Ball Joints
- Creating Rack and Pinion Joints
- Creating Screw Joints
- Creating Gear Joints
- Creating Belt Joints
- Managing Component Relationships and Assembly Motion
- Building Functional Mechanical Assemblies
- Assembly Modeling Tutorials
- Hands-on Assembly Exercises

Chapter 10: Creating 2D Engineering Drawings with TechDraw

- Understanding the TechDraw Workbench
- Creating Technical Drawing Sheets

- Using Default Drawing Templates
- Using Custom Drawing Templates
- Editing Title Block Information
- Creating 2D Drawing Views from 3D Models
- Managing Drawing View Scales
- Displaying Hidden Lines
- Creating Section Views
- Creating Simple Section Views
- Creating Complex Section Views
- Creating Detail Views
- Applying Linear and Angular Dimensions
- Applying Horizontal and Vertical Dimensions
- Applying Radius and Diameter Dimensions
- Creating Extent Dimensions
- Organizing Engineering Drawing Views
- Preparing Complete Engineering Drawing Sheets
- Exporting Technical Drawings to DXF
- Exporting Technical Drawings to SVG
- TechDraw Tutorial and Hands-on Exercise

Capstone Project: Complete Mechanical Design Workflow

- Creating a Fully Constrained Mechanical Sketch
- Developing a Parametric 3D Mechanical Component
- Applying Advanced Mechanical Features
- Creating Repeated and Mirrored Features
- Developing Multiple Mechanical Components
- Building a Functional Mechanical Assembly
- Applying Appropriate Assembly Joints
- Validating Component Relationships and Assembly Structure
- Generating a 2D Engineering Drawing

- Adding Dimensions and Section Views
- Adding Detail Views and Engineering Annotations
- Preparing the Final Technical Documentation
- Exporting the Completed Engineering Documentation
- Presenting the End-to-End FreeCAD Mechanical Design Workflow