

Autodesk **Inventor** for Mechanical Design Professional Certification

Target Audience

This course is designed for mechanical design professionals, engineers, and students who aim to gain advanced proficiency in Autodesk Inventor. It is ideal for individuals seeking Autodesk Inventor Mechanical Design Professional Certification to validate their expertise in 3D modeling, assembly creation, and detailed technical drawing.

Course Objective

To equip participants with the skills and knowledge required to efficiently create and manage complex mechanical designs using Autodesk Inventor, preparing them for professional certification and real-world design challenges.

Course Outcome

- Develop advanced skills in creating parametric 3D models, assemblies, and technical drawings.
- Master tools for design automation, simulation, and analysis to optimize mechanical components.
- Gain proficiency in working with sheet metal, weldments, and presentation files.
- Confidently prepare for and pass the Autodesk Inventor Mechanical Design Professional Certification exam.



Course Outline: The course comprises **56-hours** of theory and labs and is divided into **17** different chapters. Each chapter will be followed by hands-on lab exercises to reinforce learning and gauge understanding of the topics covered.

Chapter 1. Introduction to Inventor

Introduction to Autodesk Inventor

- Part Module
- Assembly Module
- Presentation Module
- Drawing Module
- Sheet Metal Module
- Mold Design Module

Getting Started with Autodesk Inventor

- Quick Access Toolbar

Ribbon and Tabs

- Sketch Tab
- 3D Model Tab
- Sheet Metal Tab
- Assemble Tab
- Place Views Tab
- Presentation Tab
- Tools Tab
- View Tab
- Navigation Bar
- Browser Bar
- Search Tool

Units for Dimensions

Important Terms and Their Definitions

- Feature-based Modeling
- Parametric Modeling
- Bidirectional Associativity
- Adaptive
- Design Doctor
- Constraints
- Consumed Sketch

Stress Analysis Environment

Select Other Behavior

Hotkeys

- Part Module
- Assembly Module
- Drawing Module
- Customizing Hotkeys

Creating the Sketch

Marking Menu

Color Scheme

Chapter 2. Sketching, Dimensioning, and Creating Base Features and Drawings

The Sketching Environment

- Initial Interface of Autodesk Inventor
- Starting a New File
- The Open Dialog Box
- Setting a New Project
- Import DWG

Invoking the Sketching Environment

Introduction to the Sketching Environment

Setting Up the Sketching Environment

- Modifying the Document Settings of a Sketch

Sketching Entities

Positioning Entities by Using Dynamic Input

- Drawing Lines
- Drawing Circles
- Drawing Ellipses
- Drawing Arcs
- Drawing Rectangles
- Drawing Polygons
- Drawing Slots
- Placing Points
- Creating Fillets
- Creating Chamfers

- Drawing Splines

Deleting Sketched Entities

Finishing a Sketch

Understanding the Drawing Display Tools

- Zoom All
- Zoom
- Zoom Window
- Zoom Selected
- Pan
- Orbit
- Constrained Orbit

Adding Dimensions to Sketches

- Linear Dimensioning
- Aligned Dimensioning
- Angular Dimensioning
- Diameter Dimensioning
- Radius Dimensioning
- Linear Diameter Dimensioning

Extruding the Sketches

Generating Drawing Views

Chapter 3. Adding Constraints to Sketches

Adding Geometric Constraints to a Sketch

- Perpendicular Constraint
- Parallel Constraint
- Tangent Constraint
- Coincident Constraint
- Concentric Constraint
- Collinear Constraint
- Horizontal Constraint
- Vertical Constraint
- Equal Constraint
- Fix Constraint
- Symmetric Constraint
- Smooth Constraint

Viewing the Constraints Applied to a Sketched Entity

Controlling Constraints and Applying them Automatically while Sketching

- Constraints Settings Dialog Box
- Scope of Constraint Inference

Deleting Geometric Constraints

Setting the Scale of a Sketch

Creating Driven Dimensions

Understanding the Concept of Fully-Constrained Sketches

Measuring Sketched Entities

- Measuring Distances
- Measuring the Radius of Arc or the Diameter of a Circle
- Measuring Angles
- Measuring Loops
- Measuring the Area
- Evaluating Region Properties

Chapter 4. Editing, Extruding, and Revolving the Sketches

Editing Sketched Entities

- Extending Sketched Entities
- Trimming Sketched Entities
- Splitting Sketched Entities
- Offsetting Sketched Entities
- Mirroring Sketched Entities
- Moving Sketched Entities
- Rotating Sketched Entities

Creating Patterns

- Creating Rectangular Patterns
- Creating Circular Patterns

Writing Text in the Sketching Environment

- Writing Regular Text
- Writing Text Aligned to a Geometry

Inserting Images and Documents in Sketches

Editing Sketched Entities by Dragging

Tolerances

Converting the Base Sketch into a Base Feature

Extruding the Sketch

Revolving the Sketch

Manipulators

Rotating the View of a Model in 3D Space

- Rotating the View of a Model Using the Orbit
- Changing the View Using the ViewCube
- Navigating the Model

Controlling the Display of Models

- Setting the Visual Styles
- Setting the Shadow Options
- Setting the Camera Type

Creating Freeform Shapes

Creating Predefined Solid Primitives

Chapter 5. Other Sketching and Modeling Options

Need for other Sketching Planes

Work Features

- Creating Work Planes
- Creating Work Axes
- Creating Work Points
- Other Extrusion Options
- Other Revolution Options
- The Concept of Sketch Sharing

Chapter 6. Advanced Modeling Tools-I

Advanced Modeling Tools

- Creating Holes
- Creating Fillets
- Creating Chamfers
- Mirroring Features and Models
- Creating Rectangular Patterns
- Creating Circular Patterns
- Creating Sketch Driven Patterns
- Creating Rib Features
- Thickening or Offsetting the Faces of Features
- Creating the Embossed and Engraved Features

- Applying Images on a Feature
- Assigning Different Colors/Styles to a Model
- Assigning Different Material to a Model
- Modifying the Properties of an Existing Material

Chapter 7. Editing Features and Adding Automatic Dimensions to Sketches

Concept of Editing Features

- Editing Features of a Model
- Updating Edited Features
- Editing Features Dynamically by Using 3D Grip
- Editing the Sketches of Features
- Redefining the Sketching Plane of a Sketched Feature

Suppressing and Unsuppressing the Features

Editing of a Feature Using the Direct Edit Tool

Deleting Features

Copying and Pasting Features

Manipulating Features by EOP

Adding Automatic Dimensions to Sketches

Projecting Entities in the Sketching Environment

- Projecting Edges or Faces
- Projecting Cutting Edges
- Projecting 2D Sketch on 3D Face
- Projecting DWG Geometry

Chapter 8. Advanced Modeling Tools-II

Advanced Modeling Tools

- Creating Sweep Features
- Creating Lofted Features
- Creating Coil Features
- Creating Threads
- Creating Shell Features
- Applying Drafts
- Creating Split Features

- Trimming Surfaces
- Extending Surfaces
- Deleting Faces
- Assigning Finish to Component
- Replacing Faces with Surfaces
- Creating Planar Boundary Patches
- Stitching Surfaces
- Working with the Sculpt Tool
- Working with the Bend Part Tool

Reordering the Features

Understanding the Parent-Child Relationship

Using the Sketch Doctor

Using the Design Doctor

Chapter 9. Assembly Modeling-I

Assembly Modeling

Types of Assemblies

- Top-down Assemblies
- Bottom-up Assemblies

Creating Top-down Assemblies

- Creating Components in the Assembly Module

Creating Bottom-Up Assemblies

- Placing Components in the Assembly File

Assembling Components by Using the Constrain Tool

- Assembly Tab
- Motion Tab
- Transitional Tab
- Constraint Set Tab

Specifying the Limits for Constraining

Assembling Parts by Using the Assemble Tool

Using ALT+Drag to Apply Assembly Constraints

Applying Joints to the Assembly

- Joint Tab
- Limits Tab

Showing and Hiding Relationships

- Show Relationship
- Hide Relationship
- Show Sick Relationship

Moving Individual Components
Rotating Individual Components in 3D Space

Chapter 10. Assembly Modeling-II

Editing Assembly Constraints

Editing Components

- Editing Components in the Assembly File
- Editing Components by Opening their Part Files

Creating Subassemblies

- Creating a Subassembly Using the Bottom-up Design Approach
- Creating a Subassembly Using the Top-down Design Approach

Checking Degrees of Freedom of a Component

Creating Pattern of Components in an Assembly

Replacing a Component from the Assembly File with Another Component

- Replacing a Single Instance of the Selected Component
- Replacing all Instances of the Selected Component

Mirroring Subassemblies or Components of an Assembly

Copying Subassemblies or Components of an Assembly

Deleting Components

Editing the Pattern of Components

Making a Pattern Instance Independent

Deleting Assembly Constraints

Creating Assembly Section Views in the Assembly File

- Quarter Section View
- Half Section View
- Three Quarter Section View
- Delete Section View

Analyzing Assemblies for Interference

Creating Design View Representations

- Model State Area
- Design View Area
- Positional View Area

Simulating the Motion of Components of an Assembly by Driving

- Assembly Constraint

Creating Positional Representations

Viewing the Bill of Material of the Current Assembly

Working with Assembly Features

Chapter 11. Working with Drawing Views-I

The Drawing Module

Types of Views

Generating Drawing Views

- Generating the Base View
- Generating Projected Views
- Generating Auxiliary Views
- Generating Section Views
- Generating Detail Views
- Generating Broken Views
- Generating Break Out Views
- Generating Overlay Views
- Generating Slice Views

Drafting Drawing Views

Editing Drawing Views

Deleting Drawing Views and Drawing Sheet

Moving Drawing Views

Copying Drawing Views

Rotating Drawing Views

Changing the Orientation of Drawing Views

Assigning Different Hatch Patterns to Components in Assembly Section Views

Editing the Default Hatch Style of the Sectioned Objects

Excluding Components from Assembly Section Views

Chapter 12. Working with Drawing Views-II

Modifying Drawing Standards

Inserting Additional Sheets into Drawing

Activating a Drawing Sheet

Displaying Dimensions in Drawing Views

- Retrieving Parametric Dimensions in Drawing Views
- Adding Reference Dimensions

Modifying the Model Dimensions

Editing Drawing Sheets

Creating Dimension Styles

Applying Dimension Styles

Modifying a Dimension and its Appearance Using the Shortcut Menu

Adding the Parts List

- Source Area
- BOM Settings and Properties Area
- Table Wrapping Area

Editing the Parts List

- Column Chooser
- Group Settings
- Filter Settings
- Sort
- Export
- Table Layout
- Renumber Item
- Save Item Overrides to BOM
- Member Selection
- Adding/Removing Custom Parts
- Shortcut Menu Options

Setting the Standard for the Parts List

Adding Balloons to Assembly Drawing Views

- Adding Balloons to Selected Components
- Adding Automatic Balloons

Adding Text to a Drawing Sheet

- Adding Multiline Text without a Leader
- Adding Multiline Text with Leader

Chapter 13. Presentation Module

The Presentation Module

- Inserting Assembly in the Presentation Module
- Animating an Assembly
- Tweaking Components in the Presentation Module
- Changing the Opacity of a Component
- Editing the Tweaked Components
- Creating Snapshot Views in a Presentation

- Editing Snapshot View
- Defining Units in the Presentation Files
- Creating Storyboard
- Creating Drawing Views of the Snapshot View
- Creating Video of the Presentation Files
- Creating Raster Images of the Presentation Files

Chapter 14. Working with Sheet Metal Components

The Sheet Metal Module

Setting Sheet Metal Component Parameters

- Setting the Sheet Metal Rule
- Setting the Material
- Setting the Unfolding Rule

Creating Sheet Metal Components

Folding Sheet Metal Components

Adding Flanges to Sheet Metal Components

Creating Cuts in Sheet Metal Components

Creating Seams at the Corners of Sheet Metal Components

Bending the Faces of a Sheet Metal Component

Rounding the Corners of Sheet Metal Components

Chamfering the Corners of Sheet Metal Components

Punching 3D Shapes into Sheet Metal Components

Creating Hems

Creating Contour Flanges

Creating the Flat Patterns of Sheet Metal Components

- Adding or Removing Material from the Flat Pattern

Chapter 15. Introduction to Stress Analysis

Introduction to FEA

Types of Engineering Analysis

- Structural Analysis
- Thermal Analysis
- Fluid Flow Analysis
- Electromagnetic Field Analysis
- Coupled Field Analysis

General Procedure to Conduct Finite Element Analysis

- FEA through Software

Important Terms and Definitions

Stress Analysis Using Autodesk Inventor Professional

- Creating Study
- Using the Guide Tool
- Applying Stress Analysis Settings

Study Browser Bar

Assigning Material

- Assign Material

Applying Constraints

- Fixed Constraint
- Pin Constraint
- Frictionless Constraint

Applying Loads

- Force
- Pressure
- Bearing Load
- Moment
- Gravity
- Remote Force
- Body

Meshing the Component

- Mesh View
- Mesh Settings
- Local Mesh Control
- Convergence Settings

Solution Phase of Analysis

Postprocessing the Solutions

- Generating Report
- Animating the Results

Chapter 16. Introduction to Weldments

Understanding Weldment Assemblies

Main Types of Welds in Autodesk Inventor Professional

- Cosmetic Welds
- Fillet Welds
- Groove Welds

Adding Welds to Assemblies

- Assembling the Components of Weldment Assemblies
- Preparing Assemblies for Weldments
- Adding Welds

Creating Fillet Welds

Creating Cosmetic Welds

Creating Groove Welds

Creating Symbols

Generating Report

Chapter 17. Working with Special Design Tools

Introduction

Copying the Sketches

Scaling the Sketches

Finding the Center of Gravity

Extracting the iFeature

Inserting the iFeature

Creating iMates

Applying iMates in the Assembly Environment

- Interactively place with iMates
- Automatically generate iMates on place

Viewing the iProperties

Creating User-Defined Drawing Sheets

Importing AutoCAD Blocks into Inventor

Adaptive Parts

Defining Parameters

Working with iParts

- Types of iPart Factories
- Creating iPart Factories
- Procedure to Create Standard iPart
- Procedure to Create Custom iPart
- Inserting an iPart into an Assembly
- Changing the iParts in the Assembly File

Creating 3D Sketches

- Line
- Spline
- Bend

- Include Geometry
- Intersection Curve
- Helical Curve