Autodesk Inventor for Mechanical Design Professional with Reverse Engineering

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 **64-hours** of theory and labs and is divided into **19** 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 Grips

- 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 Definition s
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

Chapter 18. Reverse Engineering from 3D Scan Data – Part I (Point Cloud, Mesh Handling & Reference Extraction)

Introduction to Reverse Engineering

Understanding Reverse Engineering
Applications in Mechanical Design
Types of 3D Scan Data

- Point Cloud Data
- Mesh Data (STL, OBJ, PLY)

Importing Scan Data into Inventor
Supported Import Formats

- STL, OBJ, PLY (Mesh)
- RCP/RCS (Point Cloud)

Importing Mesh and Point Cloud Files
Configuring Graphics Display for Large Scans
Inspecting Scan Quality in Graphics Area

Cleaning and Preparing the Scan Data
Cropping the Mesh / Point Cloud
Removing Noise and Outliers
Decimating High-Density Meshes (Reducing Triangles)
Aligning the Scan to Inventor Global Coordinate System
Orienting the Scan with Existing Geometries
Creating Datums from Scan Data
Creating Work Planes from Mesh Surfaces
Best-Fit Plane Creation from Selected Cloud Regions
Creating Work Axes from Cylindrical or Linear Features
Creating Reference Work Points from Scan Intersections
Establishing a Stable Modeling Coordinate System
Slicing the Scan for Reference Geometry
Creating Section Cuts Using Work Planes
Generating Intersected Curves from the Mesh
Using Multiple Slice Planes for Multi-Section Geometry
Extracting Key Profiles for Lofting and Surfacing
Creating 2D and 3D Sketch References
Placing 3D Points on Section Curves
Creating 3D Sketch Curves Using Scan Geometry
Controlling Spline Points for Accurate Replication
Managing Sketch Visibility and Constraints
Best Practices for Scan Data Interpretation
Identifying Functional vs Non-Functional Surfaces
Avoiding Overfitting to Irregular Scan Noise
Planning a Modeling Strategy
Breaking Complex Shapes into Rebuild Sections

Chapter 19. Reverse Engineering from 3D Scan Data – Part II (Surface Modeling, Reconstruction, Solid Conversion & Validation)
Planning the Reconstruction Strategy

Understanding Shape Complexity
Deciding Between:
• Solid Modeling
• Surface Modeling
• Hybrid Modeling
Identifying Primary Features and Split Regions
Organizing the Modeling Sequence
3D Sketch Creation for Rebuilding Geometry

Creating 3D Splines from Scan Points
Using Control Vertex and Interpolation Splines
Placing Guide Points for Loft Control
Creating Centerlines and Guide Rails
Managing Multi-Curve Relationships

Surface Modeling Tools for Reverse Engineering

Creating Boundary Surfaces
Creating Lofted Surfaces
• With Sections
• With Guide Rails
• With Centerlines
Using Sweep with Guide Rails
Trimming, Extending, and Stitching Surfaces
Fixing Gaps Between Surface Patches

Converting Surfaces into Solid Models

Thickening Surface Bodies
Stitching Multiple Surfaces into a Closed Solid
Performing Boolean Operations (Join, Cut, Intersect)
Applying Fillets, Chamfers, Ribs, and Features
Ensuring Manufacturability of Rebuilt Geometry

Refining and Finalizing the Solid Model

Adding Functional Features
Adjusting Dimensions for Accuracy
Applying Realistic Materials / Finishes
Cleaning the Feature Tree for Clarity

Deviation & Accuracy Validation

Overlaying the CAD Model with the Scan
Visual Comparison Techniques
Checking Profile Accuracy with Slices
Establishing Acceptable Tolerance Ranges
Documenting Deviation Observations

Creating Drawings and Outputs

Creating 2D Drawings for Rebuilt Models
Annotating Critical Dimensions
Saving the Final Part and Reference Scan Data
Exporting Final Outputs (STEP, IGES, DWG)