Autodesk Inventor for Mechanical Design Professional Certification 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 Objectives

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 Outcomes

- 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.
- Identify parametric modeling tools and techniques, and modify 3D models with solid and surfacing tools.
- Create and manage assemblies, and use form and mesh tools to manipulate models.
- Create and modify sheet metal designs, and use appropriate pattern tools.
- Modify parts and assemblies parametrically, and insert and position components in an assembly.
- Demonstrate the ability to create weldment assemblies.
- Place and modify views on a drawing, add annotations, and interrogate designs with view and inspect tools.

- Use and identify aspects of a project file, share material and style libraries, and manage iProperties.

Course Outline

The course comprises 64-hours of theory and labs and is divided into 16 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
- Getting Started with Autodesk Inventor
- Ribbon and Tabs
- Navigation Bar
- Browser Bar
- Search Tool
- Units for Dimensions
- Important Terms and Their Definitions
- Stress Analysis Environment
- Select Other Behavior
- Hotkeys
- Creating the Sketch
- Marking Menu
- Color Scheme
- Import Secondary File Formats
- Recognize Browser Bar Functionality
- Recognize Function of a Project File
- Share Company Standards
- Manage iProperties
- Sharing Files

Chapter 2. Sketching, Dimensioning, and Creating Base Features

- The Sketching Environment
- Invoking the Sketching Environment
- Introduction to the Sketching Environment
- Setting Up the Sketching Environment
- Sketching Entities

- Positioning Entities by Using Dynamic Input
- Deleting Sketched Entities
- Finishing a Sketch
- Understanding the Drawing Display Tools
- Adding Dimensions to Sketches
- Extruding the Sketches

Chapter 3. Adding Constraints to Sketches

- Adding Geometric Constraints to a Sketch
- Viewing the Constraints Applied to a Sketched Entity
- Controlling Constraints and Applying them Automatically while Sketching
- Deleting Geometric Constraints
- Setting the Scale of a Sketch
- Creating Driven Dimensions
- Understanding the Concept of Fully-Constrained Sketches
- Measuring Sketched Entities

Chapter 4. Editing, Extruding, and Revolving the Sketches

- Editing Sketched Entities
- Creating Patterns
- Writing Text in the Sketching Environment
- 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
- Controlling the Display of Models
- Creating Freeform Shapes
- Creating Predefined Solid Primitives
- Utilize Appropriate Modeling Strategies
- Utilize Appropriate Modeling Editing and Adjusting Strategies

Chapter 5. Other Sketching and Modeling Options

- Need for other Sketching Planes
- Work Features

Chapter 6. Advanced Modeling Tools-I

- 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
- Use and Determine Appropriate Pattern Tools

Chapter 7. Editing Features and Adding Automatic Dimensions to Sketches

- Concept of Editing Features
- 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
- Modify Parts and Assemblies Parametrically
- Use Model States for Parts

Chapter 8. Advanced Modeling Tools-II

- 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
- Creating Top-down Assemblies
- Creating Bottom-Up Assemblies
- Assembling Components by Using the Constrain Tool
- Specifying the Limits for Constraining
- Assembling Parts by Using the Assemble Tool
- Using ALT+Drag to Apply Assembly Constraints
- Applying Joints to the Assembly
- Showing and Hiding Relationships
- Moving Individual Components
- Rotating Individual Components in 3D Space
- Insert and Position Components in an Assembly

Chapter 10. Assembly Modeling-II

- Editing Assembly Constraints
- Editing Components

- Creating Subassemblies
- Checking Degrees of Freedom of a Component
- Creating Pattern of Components in an Assembly
- Replacing a Component from the Assembly File with Another 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
- Analyzing Assemblies for Interference
- Creating Design View Representations
- 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
- Leverage Model States for Assemblies, iParts, iAssemblies
- Demonstrate Knowledge of Adaptivity, Components, Relationships, and Flexibility

Chapter 11. Working with Drawing Views-I

- The Drawing Module
- Types of Views
- Generating Drawing 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
- Utilize Place 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
- 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
- Editing the Parts List
- Setting the Standard for the Parts List
- Adding Balloons to Assembly Drawing Views
- Adding Text to a Drawing Sheet
- Annotate Drawings
- Utilize View and Inspect Tabs in Parts and Assemblies

Chapter 13. 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
- 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
- Manage Sheet Metal Designs

Chapter 15. Introduction to Weldments

- Understanding Weldment Assemblies
- Main Types of Welds in Autodesk Inventor Professional
- Adding Welds to Assemblies
- Creating Fillet Welds
- Creating Cosmetic Welds
- Creating Groove Welds
- Creating Symbols
- Generating Report
- Demonstrate the Ability to Create Weldment Assemblies

Chapter 16. 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
- Viewing the iProperties

- Creating User-Defined Drawing Sheets
- Importing AutoCAD Blocks into Inventor
- Adaptive Parts
- Defining Parameters
- Working with iPart

Chapter 17. Reverse Engineering from 3D Scan Data – Part I (Point Cloud, Mesh Handling and Reference Extraction)

- Introduction to Reverse Engineering
- Understanding Reverse Engineering Applications in Mechanical Design
- Types of 3D Scan Data
- Importing Scan Data into Inventor
- Inspecting Scan Quality in Graphics Area
- Cleaning and Preparing the Scan Data
- Aligning the Scan to Inventor Global Coordinate System
- Creating Datums from Scan Data
- Establishing a Stable Modeling Coordinate System
- Slicing the Scan for Reference Geometry
- Extracting Key Profiles for Lofting and Surfacing
- Creating 2D and 3D Sketch References
- Managing Sketch Visibility and Constraints
- Best Practices for Scan Data Interpretation
- Planning a Modeling Strategy

Chapter 18. Reverse Engineering from 3D Scan Data – Part II (Surface Modeling, Reconstruction, Solid Conversion and Validation)

- Planning the Reconstruction Strategy
- Understanding Shape Complexity
- Identifying Primary Features and Split Regions
- Organizing the Modeling Sequence
- 3D Sketch Creation for Rebuilding Geometry
- Managing Multi-Curve Relationships
- Surface Modeling Tools for Reverse Engineering
- Converting Surfaces into Solid Models
- Ensuring Manufacturability of Rebuilt Geometry
- Refining and Finalizing the Solid Model
- Deviation and Accuracy Validation
- Creating Drawings and Outputs