Autodesk Inventor 2024

Course Objective

Upon completion of this Autodesk Inventor 2024 course, participants will acquire a comprehensive understanding of the software, enabling them to proficiently utilize its powerful features for 3D design, simulation, and documentation. Participants will gain hands-on experience in creating parametric models, assemblies, and detailed drawings, as well as exploring advanced functionalities such as dynamic simulation, stress analysis, and parametric design automation.

Course Outcome

By the end of the course, students will be equipped with the skills needed to efficiently design, visualize, and analyze complex engineering projects using Autodesk Inventor 2024, positioning them for success in various industries that rely on cutting-edge 3D design and engineering solutions.

Course Outline

The course comprises 40-hours of theory and labs and is divided into 15 different modules.

Chapter 1. Introduction to Autodesk Inventor

- Installing Autodesk Inventor
- Getting Started with Autodesk Inventor
- Starting a New Inventor File
- Invoking the Part Modeling Environment
- Invoking the Assembly Environment
- Invoking the Drawing Environment
- Invoking the Presentation Environment
- Identifying Inventor Files
- Creating and Accessing a Project
- Invoking a Marking Menu
- Customizing the Color Scheme
- Choosing the User Interface Theme
- Exporting Files to Other CAD Formats
- Saving Files
- Opening Existing Files

Chapter 2. Drawing Sketches with Autodesk Inventor

- Invoking the Part Modeling Environment
- Invoking the Sketching Environment
- Working with the Selection of Planes
- Specifying Units
- Specifying Grids and Snap Settings
- Creating a Line Entity
- Creating a Circle
- Creating an Ellipse
- Creating an Arc
- Creating a Rectangle
- Creating a Slot
- Creating a Polygon
- Creating a Spline
- Editing a Spline

Chapter 3. Editing and Modifying Sketches

- Trimming Sketch Entities
- Extending Sketch Entities
- Splitting Sketch Entities
- Offsetting Sketch Entities
- Creating a 2D Fillet
- Creating a 2D Chamfer
- Patterning Sketch Entities
- Creating Construction and Centerline Entities
- Mirroring Sketch Entities
- Moving a Sketch Entity
- Creating a Copy of Sketch Entities
- Rotating an Entity
- Scaling Sketch Entities
- Stretching Sketch Entities

Chapter 4. Applying Constraints and Dimensions

- Working with Constraints
- Applying Constraints
- Controlling the Display of Constraints
- Controlling Constraint Settings

- Applying Dimensions
- Controlling Dimension Settings
- Modifying/Editing Dimensions
- Working with Different States of a Sketch
- Displaying Available Degrees of Freedom

Chapter 5. Creating Base Feature of Solid Models

- Creating an Extrude Feature
- Creating a Revolve Feature
- Navigating a 3D Model in Graphics Area
- Changing the Visual Style of a Model

Chapter 6. Creating Work Features

- Creating Work Planes
- Creating Work Axes
- Creating Work Points
- Creating a User Coordinate System

Chapter 7. Advanced Modeling - I

- Using Advanced Options of the Extrude Tool
- Using Advanced Options of the Revolve Tool
- Working with a Sketch having Multiple Profiles
- Projecting Geometries
- Creating a Section View
- Editing a Feature and its Sketch
- Displaying Earlier State of a Model
- Re-ordering Features of a Model
- Measuring the Distance between Entities
- Assigning an Appearance
- Applying a Material
- Assigning Finishes/Manufacturing Data
- Calculating Physical Properties

Chapter 8. Advanced Modeling - II

- Creating a Sweep Feature
- Creating a Loft feature

- Creating a Coil feature
- Creating an Emboss Feature
- Creating a Mark Feature
- Creating a Rib Feature
- Applying an Image on a Face of a Model
- Creating a Shell Feature

Chapter 9. Patterning and Mirroring

- Creating a Rectangular Pattern
- Creating a Circular Pattern
- Creating a Sketch Driven Pattern
- Suppressing Features and Pattern Occurrences
- Unsuppressing Features and Pattern Occurrences
- Mirroring a Feature or a Body

Chapter 10. Advanced Modeling - III

- Creating Holes
- Creating Threads
- Creating Fillets
- Creating Chamfers
- Splitting a Face and a Solid Body
- Creating 3D Sketches and Curves

Chapter 11. Working with Assemblies - I

- Working with Bottom-up Assembly Approach
- Working with Top-down Assembly Approach
- Creating an Assembly using Bottom-up Approach
- Inserting Components in the Assembly Environment
- Working with Degrees of Freedom
- Applying Constraints
- Applying Joints
- Editing Constraints and Joints
- Deleting Constraints and Joints
- Moving and Rotating Individual Components

Chapter 12. Working with Assemblies - II

- Creating an Assembly by using the Top-down Approach
- Editing Assembly Components
- Patterning Assembly Components
- Mirroring Components of an Assembly
- Copying Components of an Assembly
- Creating Bill of Material (BOM) of an Assembly

Chapter 13. Creating Animation and Exploded Views

- Invoking the Presentation Environment
- Capturing Actions on the Timeline
- Capturing Tweaks on the Timeline
- Editing Time and Properties of a Tweak
- Deleting a Tweak
- Creating a Snapshot View
- Editing a Snapshot View
- Renaming a Snapshot View
- Deleting a Snapshot View
- Publishing a Snapshot View to a Raster Image
- Creating an Exploded View in a Drawing File
- Creating a New Storyboard
- Creating a New Scene
- Playing Animation of a Storyboard
- Publishing Animation to a Video File

Chapter 14. Working with Drawings

- Invoking the Drawing Environment
- Editing the Sheet Size
- Editing/Creating the Title Block
- Editing the Drafting Standard
- Creating the Base View of a Model
- Creating Projected Views
- Working with Angle of Projection
- Defining the Angle of Projection
- Creating Other Drawing Views
- Deleting a Drawing View
- Applying Dimensions

- Adding a Text/Note
- Adding a Surface Finish Symbol
- Adding a Weld Symbol
- Adding a Hole and Thread Note
- Adding the Parts List / Bill of Materials (BOM)
- Editing the Parts List / Bill of Materials (BOM)
- Adding Balloons

Chapter 15. Sheet Metal Design and Simulation

- Modify sheet metal defaults
- Create a face feature
- Create a Flange
- Create a Contour Flange
- Create a Corner Seam
- Create Punches
- Create a Bend Feature
- Create Corner Rounds
- Flat Patterns
- Starting a stress analysis
- Defining parts materials
- Constraints for your part
- Defining Input Forces
- Run the stress analysis