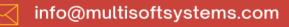# Autodesk Inventor Training

COURSE CONTENT

multisoft
Systems

About Multisoft

Train yourself with the best and develop valuable in-demand skills with Multisoft Systems. A leading certification training provider, Multisoft collaborates with top technologies to bring world-class one-on-one and certification trainings. With the goal to empower professionals and business across the globe, we offer more than 1500 training courses, which are delivered by Multisoft's global subject matter experts. We offer tailored corporate training; project Based Training, comprehensive learning solution with lifetime e-learning access, after training support and globally recognized training certificates.

About Course

The Autodesk Inventor training offered by Multisoft Systems is designed to equip participants with the skills to use Autodesk Inventor for 3D mechanical design, documentation, and product simulation.

Module 1: Autodesk Inventor Essential

1. Getting Started

- ✓ Understanding the User Interface
- ✓ Designing Parametric Parts
- ✓ Using Project Files for Part Design

2. Sketching, Constraining & Dimensioning

- ✓ Creating 2D Sketches
- ✓ Constraining the Sketches
- ✓ Adding Dimensioning to Sketches

3. Creating and Editing Placed Features

- ✓ Creating and Editing Sketched Features
- ✓ Intermediate Sketching
- ✓ Editing Parametric Parts
- ✓ Creating Work Features
- ✓ Creating Basic Swept Shapes
- ✓ Creating Chamfers and Fillets
- ✓ Creating Holes and Threads
- ✓ Patterning and Mirroring Features
- ✓ Creating Thin-Walled Parts

4. Creating and Editing Drawing Views

- ✓ Drawing Creation Environment
- ✓ Creating Styles
- ✓ Setting Drawing Standards
- ✓ Drawing Sheet Preparation
- ✓ Creating Title Blocks and Borders
- ✓ Base and Projected Views
- ✓ Section Views

- ✓ Detail Views
- ✓ Managing Views

5. Creating and Documenting Assemblies

- ✓ Designing Assemblies
- ✓ Using Project Files for Assembly Design
- ✓ Placing Existing Components in an Assembly
- ✓ Constraining Components
- ✓ Placing Standard Components Using the Content Center
- ✓ Basic Part Design in an Assembly
- ✓ Identifying Parts in an Assembly
- ✓ Analysis and Motion Tools
- ✓ Presenting Your Assembly
- ✓ Bill of Materials
- ✓ Creating and Customizing Parts Lists
- ✓ Creating Balloons

6. Dimensions, Annotations, and Tables

- ✓ Automated Dimensioning Techniques
- ✓ Manual Dimensioning Techniques
- ✓ Annotating Holes and Threads
- ✓ Creating Centerlines, Symbols, and Leaders
- ✓ Creating Tables

Module 2: Autodesk Inventor Advanced

1. Getting Started

- ✓ Design Philosophies
- ✓ Sketching, Modeling, Modification, Display Tips

2. Multi-Body Tools

- ✓ Multi-body Part Modeling

3. Advanced Work Features

- ✓ User Coordinate System
- ✓ Grounded Work Points

4. Lofts & Loft Options

- ✓ Area Lofts
- ✓ Advanced Sweeps
- ✓ Coils
- ✓ Analyzing a Model
- ✓ Analysis Types, Analysis Procedures

5. Introduction to Surfacing

- ✓ Basic Surfaces
- ✓ Patch Surfaces
- ✓ Stitch Surfaces
- ✓ Sculpting with Surfaces
- ✓ Thickening and Offsetting a Surface
- ✓ Extend and Trim Surface
- ✓ Replace Face
- ✓ Delete Face
- ✓ Copy Surface
- ✓ Importing Surfaces
- ✓ Repairing Imported Surfaces

6. Custom Sketches

- ✓ Create Sketch Symbols
- ✓ Place Sketched Symbols
- ✓ AutoCAD Blocks in Inventor

7. Engineers Notebook

- ✓ Notes

8. Effective Assembly Tools

- ✓ General Assembly Tips
- ✓ Constraint Tips
- ✓ Motion Constraints
- ✓ Transitional Constraints

9. Introduction to Top-Down Design

- ✓ Top-Down Design
- ✓ Top-Down Design Technique
- ✓ Additional Top-Down Design Tools

10. Derived Components

- ✓ Derived Component
- ✓ Modifying Derived Components

11. Assembly Parts and Features

- ✓ Assembly Parts
- ✓ Assembly Features

12. Associative Links and Adaptive Parts

- ✓ Associative Links
- ✓ Adaptive Assembly Parts

13. Skeleton Models and Derived Components

- ✓ Working with Skeletons

14. iFeatures & iParts

- ✓ Creating / Inserting / Editing iFeatures
- ✓ Table-Driven iFeatures
- ✓ Creating / Inserting / Editing iParts
- ✓ Table-Driven iParts

15. Translation

- ✓ Import and Export Data
- ✓ Import Data Formats
- ✓ Importing Solids

16. Drawing Tools

- ✓ Create Hole Tables
- ✓ Revision Tables and Tags
- ✓ Tables for Factory Members
- ✓ Surfaces in Drawing Views

17. Design Accelerator

- ✓ Design Accelerator
- ✓ Generators
- ✓ Calculators

18. Inventor Studio

- ✓ Introduction to Inventor Studio
- ✓ Rendering
- ✓ Animation
- ✓ Creating a Standard Room

19. Assemblies

- ✓ Introduction
- ✓ Create Basic assemblies
- ✓ Create multi-level assemblies
- ✓ Derived Components
- ✓ Modify Derived Components

20. Applying Motion with Constraints

- ✓ Driving Constraints
- ✓ Motion Constraints

✓ Transitional Constraints

21. Positional / Level of Detail Representations

✓ Introduction to Positional representations

✓ Create and Edit Positional representations

✓ Using a Positional Representation

✓ Level of Detail Representations

✓ System-Defined Level of Detail representations

✓ User-Defined Level of Detail Representations

✓ Using Level of Detail Representations

22. Assembly Duplication Options

✓ Pattern Components

✓ Mirror Components

✓ Copy Components

23. Assembly Bill of Materials

✓ Create Virtual Components

✓ Create Bill of Materials

✓ Frame Generator

✓ Frame Generator

✓ Structural Shape Author

Module 3: Autodesk Inventor – Sheet Metal

1. Understanding Sheet Metal Modeling

✓ Sheet Metal Concepts

✓ Sheet Metal Terminology

✓ Sheet Metal Environment

✓ Sheet Metal Design Process

✓ Working With Sheet Metal Rules

✓ Sheet, Bend and Corner Tab Options

✓ Bend Tables

2. Creating Sheet Metal Features

✓ Applying Existing Sheet Metal Defaults

✓ Creating a Face as a Base Feature

✓ Creating a Contour Flange as a Base Feature

✓ Creating a Contour Roll as a Base Feature

✓ Sheet Metal Parameters

✓ Bend Relief Shapes

✓ Faces as Secondary Features

✓ Contour Flanges as Secondary Features

✓ Contour Rolls as Secondary Features

3. Working with Flanges and Rips

✓ Creating Flanges

✓ Corner Relief Options

✓ Lofted Flanges

✓ Rips

4. Bending Sheet Metal

✓ Hems

✓ Folds

✓ Bend Features

5. Sheet Metal Cuts

✓ Creating Cut Features

✓ Creating Straight Holes

✓ Using Punch Tool Features

✓ Cuts Using Surfaces

6. Flat Pattern Environment

✓ Creating Flat Patterns

- ✓ Orienting Flat Patterns
- ✓ Punch Representations
- ✓ Bend Angle
- ✓ Flat Pattern Cleanup
- ✓ Exporting to DXF/DWG

7. Documentation

- ✓ Documentation and Annotation
- ✓ Sheet Metal Drawing Terminology
- ✓ Creating Sheet Metal Drawings
- ✓ Bend and Punch Notes
- ✓ Bend Tables
- ✓ Punch Tables
- ✓ Bend Order
- ✓ Cosmetic Centerlines

8. Converting Parts to Sheet Metal

- ✓ Converting Solid Model to Sheet Metal
- ✓ Non-Ruled Surfaces

9. Corner Rounds, Chamfers and Seams

- ✓ Creating Corner Rounds
- ✓ Creating Corner Chamfers
- ✓ Creating Corner Seams and Miters
- ✓ Creating Corner Rips
- ✓ Using Corner Seams and Bends

Module 4: Autodesk Inventor- Simulation

1. Introduction to Engineering Analysis

- ✓ Stress Analysis Overview

2. Stress Analysis

- ✓ Preparing and Running a Simulation
- ✓ Viewing Results
- ✓ Analyzing Assemblies
- ✓ Performing a Parametric Design Study
- ✓ Mesh Control and Convergence
- ✓ Performing a Modal Analysis

3. Engineering Problems and Solution

- ✓ Solving Design Problems

Module 5: Autodesk Inventor Associate Preparation

1. User Interface

- ✓ Describe how to use the heads-up display (HUD) to create and edit features
- ✓ Identify how to use visual styles to control the appearance of a model

2. Project Files

- ✓ Describe the options for controlling a project file

3. Sketching

- ✓ Recall the function of each sketch constraint
- ✓ Demonstrate how to create dynamic input dimensions

4. Part Modeling

- ✓ Create extrude features
- ✓ Create fillet features
- ✓ Create hole features
- ✓ Create a pattern of features
- ✓ Describe how to use the Project Geometry and Project Cut Edges commands
- ✓ Create revolve features
- ✓ Create a shell feature

✓ Create work features and a UCS

5. Drawing

✓ Explain how to edit a base and

✓ projected views

✓ Describe how to create a slice view in a drawing

✓ Demonstrate how to create and edit dimensions in a drawing

✓ Describe how to edit a hole table

✓ Describe how to modify a parts list

✓ Demonstrate how to edit a section view

6. Assembly Modeling

✓ Describe the process of finding the minimum distance between parts and
components

✓ Describe the function of the different assembly constraints

✓ Describe how to modify a bill of materials

✓ Explain the method of creating a frame using the frame generator command

✓ Identify uses for surfaces in the modeling process

7. Presentation Files

✓ Describe how to animate a presentation file

8. Advanced Modeling

✓ Describe the process to emboss text and a profile

✓ Create and constrain sketch blocks

✓ Describe the process of creating an Assembly

✓ Describe the process to create an iPart

9. Sheet Metal

✓ Demonstrate how to create and edit a sheet metal flat pattern

✓ Describe the different types of sheet metal flanges that Inventor can create

✓ Demonstrate how to annotate a sheet metal part in a drawing

Module 6: Autodesk Inventor Professional Preparation

1. Part Modeling

- ✓ Create extrude features
- ✓ Create hole features

2. Drawing

- ✓ Demonstrate how to edit a section view
- ✓ Create a slice view in a drawing
- ✓ Demonstrate how to modify a style in a drawing

3. Assembly Modeling

- ✓ Apply assembly constraints
- ✓ Create a part in the context of an assembly
- ✓ Create components using the Design
- ✓ Accelerator commands
- ✓ Create and edit a frame using the Frame
- ✓ Generator command
- ✓ Create a level of detail
- ✓ Create a positional representation

4. Advanced Modeling

- ✓ Create a 3D path using the Intersection Curve and the Project to Surface commands
- ✓ Create a multi-body part
- ✓ Create a part using surfaces
- ✓ Create an iPart
- ✓ Create a loft feature
- ✓ Create plastic part features
- ✓ Create a sweep feature

5. Sheet Metal

✓ Create flanges using the Flange, Contour Flange, and Lofted Flange commands

6. Weldments

✓ Create a weldment