

## **Autodesk Inventor Professional Introduction for Experienced 3D CAD Users**

**Course Length: 3 days**

The Autodesk Inventor Introduction for Experienced 3D CAD Users training course is intended to provide accelerated introductory training in the Autodesk® Inventor® software. This learning guide is designed for users that have 3D modelling design experience with other 3D CAD software packages (e.g., CATIA™, Pro/ENGINEER®, Creo Parametric™, NX™, SolidWorks®, etc.).

By leveraging the experience users gain in working with other 3D modelling software packages, this hands-on, practice-intensive training course is developed so that new users in the Autodesk Inventor software can benefit from a shorter, introductory-level, training course. You are taught how to find and use the modelling tools associated with familiar modelling strategies that are used in other 3D CAD software. You will acquire the knowledge required to complete the process of creating models from conceptual sketching, through to solid modelling, assembly design, and drawing production.

### **Topics Covered**

- The Autodesk Inventor software interface
- Obtaining model information
- Creating sketch and pick and place features
- Work Features
- Creating equations and working with parameters
- Model geometry and model display manipulation
- Feature duplication techniques
- Placing and constraining parts in assemblies
- Assembly component display
- Presentation files (Exploded views and Animations)
- Assembly tools
- Creating parts and features in assemblies
- Creating and editing assembly Bill of Materials
- Working with projects
- Creating and annotating drawings and views

### **Prerequisites**

Prior knowledge of 3D modelling and 3D CAD software. Users with AutoCAD® or AutoCAD® Mechanical experience are recommended to use the Autodesk Inventor Introduction to Solid Modelling course.

## Training Guide Contents

### Chapter 1: Introduction to Autodesk Inventor

- 1.1 Getting Started
- 1.2 Autodesk Inventor Interface
- 1.3 Model Manipulation
- 1.4 Model Information

### Chapter 2: Sketching Geometry

- 2.1 Creating a New Part File
- 2.2 Creating a Sketch
- 2.3 Sketch Geometry
- 2.4 Constraints
- 2.5 Dimensions
- 2.6 Additional Sketching Tools

### Chapter 3: Creating Sketched Features

- 3.1 Extrude & Revolve Features
- 3.2 Sweep Features
- 3.3 Loft Features
- 3.4 Editing Sketched Features

### Chapter 4: Creating Pick and Place Features

- 4.1 Edge Chamfer
- 4.2 Constant Fillets
- 4.3 Variable Fillets
- 4.4 Face Fillets
- 4.5 Full Round Fillets
- 4.6 Holes
- 4.7 Threads
- 4.8 Editing Pick and Place Features

### Chapter 5: Work Features

- 5.1 Work Planes
- 5.2 Work Axes
- 5.3 Work Points

### Chapter 6: Additional Features

- 6.1 Face Draft
- 6.2 Splitting a Face or Part
- 6.3 Shells
- 6.4 Ribs
- 6.5 Reordering Features

- 6.6 Inserting Features

## **Chapter 7: Equations**

- 7.1 Equations
- 7.2 Parameters

## **Chapter 8: Duplication Tools**

- 8.1 Rectangular Sketch Patterns
- 8.2 Circular Sketch Patterns
- 8.3 Rectangular Feature Patterns
- 8.4 Circular Feature Patterns
- 8.5 Sketched Driven Patterns
- 8.6 Mirror Parts or Features
- 8.7 Manipulate Patterns and Mirror Features

## **Chapter 9: Assembly Environment**

- 9.1 Assembling Components Using Constraints
- 9.2 Assemble Mini-Toolbar
- 9.3 Content Centre
- 9.4 Assembly Browser
- 9.5 Saving Files

## **Chapter 10: Joint Connections**

- 10.1 Assembling Components Using Joints

## **Chapter 11: Manipulating Assembly Display**

- 11.1 Moving and Rotating Assembly Components
- 11.2 Suppressing Constraints
- 11.3 Component Display
- 11.4 Selection Options in Assemblies

## **Chapter 12: Presentation Files**

- 12.1 Creating Presentations
- 12.2 Storyboards
- 12.3 Snapshot Views
- 12.4 Publishing a Presentation File

## **Chapter 13: Assembly Tools**

- 13.1 Replacing Components
- 13.2 Duplicating Components
- 13.3 Restructuring Components

- 13.4 Driving Constraints
- 13.5 Contact Solver
- 13.6 Interference
- 13.7 Error Recovery

## **Chapter 14: Assembly Parts and Features**

- 14.1 Assembly Parts
- 14.2 Assembly Features

## **Chapter 15: Assembly Bill of Materials**

- 15.1 Create Virtual Components
- 15.2 Create Bill of Materials

## **Chapter 16: Working With Projects**

- 16.1 Project Files
- 16.2 Resolving Links

## **Chapter 17: Drawing Basics**

- 17.1 Creating a New Drawing
- 17.2 Base and Projected Views
- 17.3 Additional Drawing Views
- 17.4 Manipulating Views

## **Chapter 18: Detailing Drawings**

- 18.1 Dimensions
- 18.2 Drawing Sheets
- 18.3 Parts List
- 18.4 Balloons
- 18.5 Styles and Standards
- 18.6 Hatching

## **Chapter 19: Drawing Annotations**

- 19.1 Text
- 19.2 Symbols
- 19.3 Hole and Thread Notes
- 19.4 Chamfer Notes
- 19.5 Centre Marks and Centre Lines
- 19.6 Hole Tables
- 19.7 Revision Tables and Tags

## **Appendix A: Sketching Options**

- A.1 Sketch Geometry Creation Options
- A.2 Sketch Editing Options
- A.3 Sketch Constraint Options

- A.4 Dimension Type Options

## **Appendix B: Customizing Autodesk Inventor**

- B.1 Application Options
- B.2 Document Settings
- B.3 File Properties
- B.4 Changing Part Units
- B.5 Command Customization

## **Appendix C: Additional Practices Appendix D: Autodesk Inventor Certification Exam Objectives**