

## Autodesk Inventor for Experienced 3D CAD Users

The Autodesk Inventor for Experienced 3D CAD Users student guide is intended to provide accelerated introductory training in the Autodesk Inventor software. This student guide can be used in a 3-day introductory training class and is designed for users that have 3D modeling design experience with other 3D CAD software packages (e.g., CATIA, Pro/ENGINEER, Creo Parametric, NX, SolidWorks, etc.).

## Objectives

The new user is taught how to find and use the modeling tools associated with familiar modeling strategies that are used in other 3D CAD software. Users of this student guide acquire the knowledge required to complete the process of creating models from conceptual sketching, through to solid modeling, assembly design, and drawing production.

## Duration

3 days

## Prerequisites

As an accelerated introductory student guide, Autodesk Inventor for Experienced 3D CAD Users assumes prior knowledge of 3D modeling and other 3D CAD software.

## Topics Covered

- Introduction to Autodesk Inventor
  - Getting Started
  - Autodesk Inventor Interface
  - Model Manipulation
  - Model Information
- Sketching Geometry
  - Creating a New Part File
  - Creating a Sketch
  - Sketch Geometry
  - Constraints
  - Dimensions
  - Additional Sketching Tools
- Creating Sketched Features
  - Extrude Features
  - Revolve Features
  - Sweep Features
  - Loft Feature
  - Editing Sketched Features
- Creating Pick and Place Features
  - Edge Chamfer
  - Constant Fillets
  - Face Fillets
  - Full Round Fillets
  - Straight Holes
  - Threads
  - Editing Pick and Place Features
- Work Features
  - Work Planes
  - Work Axes
  - Work Points
- Additional Features
  - Face Draft
  - Splitting a Face or Part
  - Shells
  - Ribs
  - Reordering Features
- Inserting Features
- Equations
  - Model Parameters
  - User Parameters
- Duplication Tools
  - Rectangular Sketch Patterns
  - Circular Sketch Patterns
  - Rectangular Feature Patterns
  - Circular Feature Patterns
  - Sketched Driven Patterns
  - Mirror Parts or Features
  - Manipulate Patterns and Mirror Features
- Assembly Environment
  - Assembling Components Using Constraints
  - Assemble Mini-Toolbar
  - Content Center
  - Assembly Browser
  - Saving Files
- Assembling Components Using Joints
- Manipulating Assembly Display
  - Moving and Rotating Assembly Components
  - Suppressing Constraints
  - Component Display
  - Selection Options in Assemblies
- Presentation Files
  - Creating Presentations
  - Storyboards
  - Snapshot Views
  - Publishing a Presentation File
- Assembly Tools

- Replacing Components
- Duplicating Components
- Restructuring Components
- Driving Constraints
- Contact Solver
- Interference
- Error Recovery
- Assembly Parts and Features
  - Assembly Parts
  - Assembly Features
- Assembly Bill of Materials
  - Create Virtual Components
  - Create Bill of Materials
- Working With Projects
  - Project Files
  - Resolving Links
- Drawing Basics
  - Creating a New Drawing
  - Base and Projected Views
  - Additional Drawing Views
  - Manipulating Views
- Detailing Drawings
  - Dimensions
  - Drawing Sheets
  - Parts List
  - Balloons
  - Styles and Standards
  - Hatching
- Drawing Annotations
  - Text
  - Symbols
  - Hole and Thread Notes
  - Chamfer Notes
  - Center Marks and Center Lines
  - Hole Tables
  - Revision Tables and Tags