Autodesk Inventor Advanced

Overview

The Inventor Advanced course will explain the Autodesk Inventor settings while teaching you how each tool functions. Just as importantly, though, the course content is filled with the tips and techniques learned by the experts who spent years using, researching, and discussing the tools in Autodesk Inventor. You should come away from this course with a solid understanding of the capabilities of Autodesk Inventor and a strong idea of how to tackle your design challenges in the future, as well as an abundance of time-saving tips and tricks.

Prerequisites

It is recommended that delegates have a working knowledge of one or more of the following:

- [Autodesk Inventor Essentials](#) (Click for More).
- At least 1 Month hands on practice / working experience while using the software daily.
- Parametric part and assembly design using Autodesk Inventor.
- Parametric solid modelling

Available Exams and Certifications

- [Autodesk Certified User](#) (Click for More)
- [Autodesk Certified Professional](#) (Click for More)

Course Outline

Sketch Techniques

- Exploring the Options and Settings for Sketches
- Sketching Basics
- Creating a Sketch in a New Part
- Taking a Closer Look at Sketch Constraints
- Gaining More Sketch Skills
- Creating Sketches from AutoCAD Geometry
- Creating and Using 3D Sketches
- Best Practices for Working with Sketches

Advanced Modeling Techniques

- Creating Complex Sweeps and Lofts
- Creating Multi-Body Parts
- Creating Derived Parts and Assemblies
- Working with Patterns
- Setting iProperties and Parameters
- Adding Part Tolerances
- Troubleshooting Failures with the End-of-Part Marker

Sheet Metal

- Understanding Sheet-Metal Parts
- Getting to Know the Features

Using Sheet-Metal Templates and Rule

- Working with the Flat Pattern
- Using Sheet-Metal iPart Factories
- Modeling with Non-Sheet-Metal Features
- Working with Imported Parts
- Annotating Your Sheet-Metal Design

Reusing Parts and Features

- Working with iParts
- Working with iFeatures
- Reusing Existing Geometry

Assembly Design Workflows

- Assembly Relationships
- Understanding Subassemblies
- Top-Down Design
- Adaptivity
- Assembly Features
- Managing the Bill of Materials
- Assembly Reuse and Configurations
- Use Assembly Design Accelerators
- Simplifying Parts

Large Assembly Strategies

- Selecting a Workstation
- Working with Performance Settings
- Large Assembly Best Practices
- Managing Assembly Detail

Weldment Design

- Exploring Weldment Design Methodologies
- Modeling Preparations
- Exploring Cosmetic Welds
- Creating Weld Beads
- Creating Fillet Welds
- Creating Groove Welds
- Performing Machining Operations
- Exploring Weld Properties and Combinations
- Using the Weld Symbol
- Understanding Bead Property Report and Mass Properties
- Creating Drawing Documentation
- Generating a Bill of Materials and Parts List

Presentations and Exploded Views

- Working in the Presentation Environment
- Creating Advanced Presentations
- Creating and Sharing Assembly Instructions

Documentation

- Using the Drawing Manager
- Creating Templates and Styles
- Utilizing Drawing Resources
- Editing Styles and Standards
- Creating Drawing Views
- Annotating Part Drawings
- Annotating Assembly Drawings
- Working with Sheet-Metal Drawings
- Working with Weldment Views
- Working with iParts and iAssembly Drawings
- Sharing Your Drawings outside Your Workgroup

Tools Overview

- Exploring the BIM Exchange
- Using the Design Assistant
- Using the Drawing Resource Transfer Wizard
- Using the Style Library Manager.
- Using the Task Scheduler

- Using iProperties
- Using the Measure Tools
- Using Miscellaneous Tools
- Exchanging Data with Other Systems
- Importing and Exporting Geometry
- Using Inventor File Translators
- Working with Imported Data and Using Inventor Fusion
- Viewing DWF Markup

Frame Generator

- Accessing Frame Generator Tools
- Exploring the Anatomy of a Frame Member
- Inserting Frame Members
- Adding End Treatments
- Maintaining Frames
- Performing Calculations and Analysis
- Publishing Frame Members
- Frame Assemblies and BOMs

Inventor Studio

- Exploring the Inventor Studio Environment
- Creating and Managing Styles
- Animating with Inventor Studio

Stress Analysis and Dynamic Simulation

- Introduction to Analysis
- Conducting Stress Analysis Simulations
- Conducting Dynamic Simulations
- Exporting to FEA
- Using the Dynamic Simulation Information in Stress Analysis

Routed Systems

- Tube and Pipe
- Cable and Harness

Plastic Design Features

- Creating Thicken/Offset Features
- Creating Shell Features
- Creating Split Features
- Creating Grill Features
- Creating Rule Fillet Features
- Creating Rest Features
- Creating Boss Features
- Creating Lip and Groove Features

- Creating Snap Fit Features
- Creating Rib and Web Features
- Creating Draft Features
- Mold Design Overview

iLogic

- What is iLogic?
- Understanding the iLogic Elements and Interface
- Creating iLogic Parameters, Rules, and Forms