Autodesk Inventor: Introduction to Solid Modeling

Course Length: 5 Days

The Autodesk Inventor: Introduction to Solid Modeling training course provides you with an understanding of the parametric design philosophy through a hands-on, practice-intensive curriculum. You will learn the key skills and knowledge required to design models using Autodesk Inventor, starting with conceptual sketching, through to solid modeling, assembly design, and drawing production.

Topics Covered

- Understanding the Autodesk Inventor software interface
- Creating, constraining, and dimensioning 2D sketches
- Creating and editing the solid base 3D feature from a sketch
- Creating and editing secondary solid features that are sketched and placed
- Creating equations and working with parameters
- Manipulating the display of the model
- Resolving feature failures
- Duplicating geometry in the model
- Placing and constraining/connecting parts in assemblies
- Manipulating the display of components in an assembly
- Obtaining model measurements and property information
- Creating presentation files (exploded views)
- Modifying and analyzing the components in an assembly
- Simulating motion in an assembly
- Creating parts and features in assemblies
- Creating and editing an assembly bill of materials
- Working with projects

Course description shown for Autodesk Inventor 2022. Topics, curriculum, and/or prerequisites may change depending on software version.

- Creating and annotating drawings and views
- Customizing the Autodesk Inventor environment

Prerequisites

As an introductory course, Autodesk Inventor: Introduction to Solid Modeling does not assume prior knowledge of any 3D modeling or CAD software. You need to be experienced with the Windows operating system, and having a background in drafting of 3D parts is recommended.

Learning Guide Contents

Chapter 1: Introduction to Autodesk Inventor

- 1.1 Autodesk Inventor Introduction
- 1.2 Getting Started with Inventor
- 1.3 Autodesk Inventor Interface
- 1.4 Model Orientation
- 1.5 Model Display
- 1.6 Selection Techniques

Chapter 2: Creating the Base Feature

- 2.1 Creating a New Part File
- 2.2 Sketched Base Features
- 2.3 Editing Sketched Features

Chapter 3: Additional Sketching Tools

- 3.1 Advanced Sketched Entities
- 3.2 Basic Sketch Editing Tools
- 3.3 Adding and Modifying Sketch Constraints
- 3.4 Advanced Dimensioning Techniques

Chapter 4: Sketch Editing Tools

- 4.1 Advanced Sketch Editing Tools
- 4.2 Rectangular Sketch Patterns
- 4.3 Circular Sketch Patterns
- 4.4 Sketch Preferences

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Chapter 5: Sketched Secondary Features

- 5.1 Creating Sketched Secondary Features
- 5.2 Offsetting Sketch Geometry
- 5.3 Projecting Geometry
- 5.4 Sharing a Sketch
- 5.5 Sketching with Dynamic and Precise Input
- 5.6 Using AutoCAD Data in Inventor

Chapter 6: Creating Pick and Place Features

- 6.1 Edge Chamfers
- 6.2 Constant Fillets
- 6.3 Variable Fillets
- 6.4 Face Fillets
- 6.5 Full Round Fillets
- 6.6 Holes
- 6.7 Threads
- 6.8 Editing Pick and Place Features
- 6.9 Creation Sequence

Chapter 7: Work Features

- 7.1 Work Planes
- 7.2 Work Axes
- 7.3 Work Points

Chapter 8: Equations and Parameters

- 8.1 Creating Equations
- 8.2 Model and User Parameters

Chapter 9: Additional Features

- 9.1 Creating Drafts
- 9.2 Splitting a Face or Solid
- 9.3 Shells
- 9.4 Ribs

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Chapter 10: Model and Display Manipulation

- 10.1 Reordering Features
- 10.2 Inserting Features
- 10.3 Suppressing Features
- 10.4 Sectioning Part Models
- 10.5 Part Design Views

Chapter 11: Fixing Problems

- 11.1 Sketch Failure
- 11.2 Feature Failure

Chapter 12: Sweep Features

- 12.1 Sweep Features

Chapter 13: Loft Features

- 13.1 Creating Rail and Center Line Lofts
- 13.2 Setting Conditions and Transitions for Lofts

Chapter 14: Duplication Tools

- 14.1 Rectangular Feature Patterns
- 14.2 Circular Feature Patterns
- 14.3 Sketched Driven Patterns
- 14.4 Mirror Features or Solids
- 14.5 Manipulating Patterns

Chapter 15: Feature Relationships

- 15.1 Establishing Feature Relationships
- 15.2 Controlling Feature Relationships
- 15.3 Investigating Feature Relationships
- 15.4 Changing Feature Relationships

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Chapter 16: Assembly Environment

- 16.1 Assembling Components Using Constraints
- 16.2 Assemble Mini-Toolbar
- 16.3 Content Center
- 16.4 Assembly Browser
- 16.5 Saving Assembly Files

Chapter 17: Joint Connections

- 17.1 Assembling Components Using Joints

Chapter 18: Manipulating Assembly Display

- 18.1 Moving and Rotating Assembly Components
- 18.2 Suppressing Constraints
- 18.3 Controlling Assembly Component Display
- 18.4 Sectioning Assembly Models
- 18.5 Assembly Design Views
- 18.6 Assembly Selection Filters

Chapter 19: Model Information

- 19.1 Measurement Tools
- 19.2 Model Material and Appearance Settings

Chapter 20: Presentation Files

- 20.1 Presentation Files - Getting Started
- 20.2 Presentation Files - Storyboard Animations
- 20.3 Presentation Files - Snapshot Views
- 20.4 Presentation - Publishing Presentation Files

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Chapter 21: Assembly Tools

- 21.1 Replacing Components
- 21.2 Duplicating Components
- 21.3 Restructuring Components
- 21.4 Driving Constraints
- 21.5 Contact Solver
- 21.6 Interference Detection
- 21.7 Error Recovery

Chapter 22: Assembly Parts and Features

- 22.1 Creating Parts in an Assembly
- 22.2 Creating Assembly Features

Chapter 23: Assembly Bill of Materials

- 23.1 Create Virtual Components
- 23.2 Create Bill of Materials
- 23.3 Instance Properties in a BOM

Chapter 24: Working with Projects

- 24.1 Project Files
- 24.2 Resolving Links

Chapter 25: Drawing Basics

- 25.1 Creating a New Drawing
- 25.2 Base and Projected Views
- 25.3 Additional Drawing Views
- 25.4 Manipulating Views

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Chapter 26: Detailing Drawings

- 26.1 Adding Dimensions to Drawing Views
- 26.2 Drawing Sheets
- 26.3 Parts List
- 26.4 Drawing Balloons
- 26.5 Styles and Standards
- 26.6 Drawing View Hatching

Chapter 27: Drawing Annotations

- 27.1 Drawing Text
- 27.2 Symbols
- 27.3 Hole and Thread Notes
- 27.4 Chamfer Notes
- 27.5 Center Marks and Center Lines
- 27.6 Hole Tables
- 27.7 Revision Tables and Tags

Chapter 28: Customizing Autodesk Inventor

- 28.1 Application Options
- 28.2 Document Settings
- 28.3 File Properties
- 28.4 Changing Part Units
- 28.5 Inventor User Interface Customization

Appendix A: Sketching Options

- A.1 Summary of the Sketch Geometry Creation Options
- A.2 Summary of the Sketch Editing Options
- A.3 Summary of the Sketch Constraint Options
- A.4 Dimension Type Options

Appendix B: Primitive Base Features

- B.1 Primitive Base Features

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- For Customer cancellations when written notice is received (i) at least ten (10) business days in advance of the class, the Customer is entitled to a full refund of its payment or reschedule enrollment, (ii) less than ten (10) business days, Customer shall not be entitled to a refund, but shall receive a class credit to be used within three (3) months of the date of the original class.
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