



Autodesk Inventor 2010 Essentials

Students learn the fundamental principles of 3D parametric part design, assembly design, and creating production-ready part and assembly drawings using Autodesk® Inventor™. Hands-on exercises representing real-world, industry-specific design scenarios are included.

Objectives

The primary objective of this courseware is to provide students with a thorough understanding of the principal 3D design, validation, and documentation processes necessary for developing products using Autodesk Inventor.

After completing this course, students will be able to:

- Identify the main user interface components that are common to all Autodesk Inventor design environments and describe how to access different tools. Describe the characteristics and benefits of a parametric part model and how to view all aspects of your design by efficiently navigating around in 2D and 3D space.
- Use sketch tools to create 2D sketch geometry, apply geometric constraints to control sketch geometry, and add parametric dimensions to your sketch geometry.
- Create features using the Extrude and Revolve tools, use reference and construction geometry, use the browser and shortcut menus to edit parametric parts, use the 3D Grips tool to edit part geometry in the context of an assembly and in a stand-alone part, create, locate, and utilize work features to perform modeling tasks, and create swept shapes by sweeping a profile along a 2D or 3D path.
- Create both chamfers and fillets on a part, use the Hole and Thread tools to place hole and thread features, create rectangular and circular patterns and mirror existing features, and create thin walled parts.
- Describe the assembly modeling process, the Autodesk Inventor assembly modeling environment, and recommended assembly design workflows, and how to use Autodesk Inventor project files to manage design projects.
- Place components in an assembly, add constraints to components, use Content Center to place standard components in an assembly, and create new components in the context of an assembly.
- Use different tools and methods to identify, locate, and select components in an assembly, retrieve important analysis information from the parametric models and assemblies, and create animations of exploded views in a presentation file
- Navigate the Autodesk Inventor user interface when creating and editing drawing sheets, create base and projected views of 3D parts and assemblies, create and edit section views, detail views, and cropped views, and manage drawing views.
- Dimension drawings with automated and manual techniques, create and edit hole and thread notes in drawings, add centerlines, center marks and symbols to your drawings, and configure, add, and edit revision tables and revision tags.
- View and edit bill of materials data, create and customize parts lists to document the components in your assembly, and review balloons and their purpose in the drawing annotation process.

Duration

4 days

Who should attend?

This courseware is designed for new Autodesk Inventor users.

Typical Schedule

Unless otherwise noted on your class registration e-mail, this class starts each day at 9:00 am and ends at 4:00 pm.

Prerequisites

No previous CAD experience is necessary. However, before using this courseware, the student should have a working knowledge of the following:

- Drafting, design, or mechanical engineering principles.
- Microsoft® Windows® Vista, Microsoft® Windows® XP, or Microsoft® Windows® 2000.

Outline

Getting Started

- Autodesk Inventor User Interface
- View Manipulation
- Designing Parametric Parts

Basic Sketching Techniques

- Creating 2D Sketches
- Geometric Constraints
- Dimensioning Sketches

Basic Shape Design

- Creating Basic Sketched Features
- Intermediate Sketching
- Editing Parametric Parts
- 3D Grip Editing
- Creating Work Features
- Creating Basic Swept Shapes

Detailed Shape Design

- Creating Chamfers and Fillets
- Creating Holes and Threads

- Patterning and Mirroring Features
- Creating Thin-Walled Parts

Assembly Design Overview

- Designing Assemblies
- Using Project Files in Assembly Designs

Placing, Creating, and Constraining Components

- Placing Components in an Assembly
- Constraining Components
- Placing Standard Components Using the Content Center
- Basic Part Design in an Assembly

Interacting with an Assembly

- Identifying Parts in an Assembly
- Analysis and Motion Tools
- Presenting Your Assembly

Basic View Creation

- Drawing Creation Environment
- Base and Projected Views

- Section Views
- Cropped Views
- Detail Views
- Managing Views

Dimensions, Annotations, and Tables

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

Annotating Assembly Drawings

- Assembly-centric Bill of Materials
- Creating and Customizing Parts Lists
- Creating Balloons

Drawing Standards and Resources

- Setting Drawing Standards
- Drawing Resources

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