Autodesk Inventor 11 - Essentials

Course description

This course covers the fundamental principles of 3D parametric part design, assembly design, and creating production-ready part and assembly drawings using Autodesk Inventor®. Students learn how to:

- Capture design intent by learning the proper techniques and recommended workflows for creating intelligent 3D parametric parts.
- Create, place and constrain custom and standard components in an assembly.
- Simulate mechanisms, animate assembly designs, and check for interferences.

Students also learn how to document their designs using base, projected, section, detail, and isometric drawings views, document assemblies using standard and exploded drawing views, and follow drafting standards while dimensioning and annotating drawing views with automated balloons and part lists.

Hands-on exercises representing real-world, industry-specific design scenarios are included.

Course duration: 4 days
Courseware provided: Volume 1 - 424; Volume 2 - 406 pages
Trial CD: Yes

Objectives

The primary objective of this course is to provide students with a thorough understanding of the principal 3D design, validation, and documentation processes that they will use to develop products using Autodesk Inventor. Upon completion of the course, the student will be proficient in 3D part and assembly design and documenting those designs using part and assembly drawing creation and annotation techniques.

Who should attend?

This course is for new Autodesk Inventor users.

Pre-requisites

No previous CAD experience is necessary. However, drafting, design, or mechanical engineering experience is a plus. It is recommended that the student have a working knowledge of Microsoft® Windows® XP or Microsoft® Windows® 2000.

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Course Outline

Day 1

Parametric Part Design and Basic Sketching

- Autodesk Inventor User Interface
- Designing Parametric Parts
- Creating 2D Sketches
- Geometric Constraints
- Dimensioning Sketches

Basic Shape Design

- Creating Basic Sketched Features
- Intermediate Sketching
- Editing Parametric Parts
- Creating Work Features
- Creating Basic Swept Shapes
- Creating Basic Blended Shapes

Detailed Shape Design

- Creating Chamfers and Fillets
- Creating Holes and Threads
- Paterning and Mirroring Features
- Creating Thin-Walled Parts
- Strengthening Parts with Ribs and Webs

Day 2

Assembly Design Overview

- Designing Assemblies
- Using Project Files for Assembly Design

Placing, Creating, and Constraining Components

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

Basic Assembly Tools

- Identifying Parts in an Assembly
- Analysis and Motion Tools

Day 3

Basic View Creation

- Drawing Creation Environment
- Base and Projected Views
- Section Views
- Detail Views
- Broken Views
- Managing Views

Dimensions, Annotations, and Tables

- Automated Dimsioning Techniques
- Manual Dimensioning Techniques
- Holes and Thread Notes
- Hole Tables
- Creating Centerlines, Symbols, and Leaders
- Creating Tables

Day 4

Preparing Assemblies for Drawings

- View Representations
- Positioinal Representations
- Level of Detail Representations

Creating Assembly Drawing Views

- Creating Custom Assembly Views
- Break-out Section Views
- Overlay Views
- Creating Exploded Drawing Views
- Presenting Your Assembly

Annotating Assembly Drawings

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

Drawing Standards and Resources

- Setting Drawing Standards
- Drawing Resources

* Course topics may be modified by the instructor based upon the knowledge and skill level of the course participants.

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