

Autodesk Inventor Essentials Training

Length: 3 Days

Summary: Autodesk Inventor Essentials Training is designed for new and beginner Autodesk Inventor users who want to learn the essential tool and principles of 3D parametric part design, assembly design and creating production-ready part assembly drawings using Autodesk Inventor. The course is designed to give students an excellent foundation to understand and familiarize with the features and commands of Inventor.

Objective:

- Utilize the Design Support system and on line help
- Create, Constrain, and Edit sketched features
- Effectively work with Construction Planes, Axes and Points
- Fillets, Chamfers, Shells and Patterns
- Create Views, Sheets, Title blocks, and edit part drawings
- Use Assembly modeling to Create, Place and Constrain components
- Design and sketch adaptive parts with features and occurrences
- Manage model data using Templates and Design Assistant
- Exchange model data by Linking, Translating and Importing

Prerequisite:

- A working knowledge of basic design/drafting procedures and terminology.
- Working knowledge of Microsoft® Windows®.
- Newcomers to Autodesk Inventor
- Users who have previously used a different CAD tool & are looking to migrate to Autodesk Inventor

COURSE CONTENT

DAY 1

CHAPTER 1: INTRODUCTION TO AUTODESK INVENTOR

- Autodesk Inventor Fundamentals
- Autodesk Inventor Interface
- Model Manipulation
- Chapter exercise

CHAPTER 2: CREATING THE BASE FEATURE

- Creating a New Part File
- Sketched Base Features

- Primitive Base Features
- Chapter exercise

CHAPTER 3: SKETCHING GEOMETRY

- Sketch Geometry
- Chapter exercise

CHAPTER 4: ADDITIONAL SKETCHING TOOLS

- Advanced Editing Tools
- Rectangular Sketch Patterns
- Circular Sketch Patterns

- Over-Dimensioned Sketches
- Sketch Preferences
- Chapter exercise

CHAPTER 5: SKETCHED SECONDARY FEATURES

- Extruded Secondary Features
- Revolved Secondary Features
- Using Existing Geometry
- Editing Sketched Secondary Features
- 3D Grip Modification
- Chapter exercise

CHAPTER 6: CREATING PICK AND PLACE FEATURES

- Edge Chamfer
- Constant Fillets
- Variable Fillets
- Face Fillets
- Full Round Fillets
- Straight Holes
- Threads
- Editing Pick and Place Features
- Creation Sequence
- Chapter exercise

CHAPTER 7: WORK FEATURES

- Work Planes
- Work Axes
- Work Points
- Chapter exercise

CHAPTER 8: EQUATIONS

- Equations
- Parameters
- Chapter exercise

CHAPTER 9: ADDITIONAL FEATURES

- Face Draft
- Splitting a Face or Part
- Shells
- Ribs
- Bend Part
- Chapter exercise

CHAPTER 10: MODEL AND DISPLAY MANIPULATION

- Reordering Features
- Inserting Features
- Suppressing Features
- Section Views
- Design Views
- Chapter exercise

DAY 2

CHAPTER 11: FIXING PROBLEMS

- Sketch Failure
- Feature Failure
- Chapter exercise

CHAPTER 12: SWEEP FEATURES

- Sweep Features
- Chapter exercise


CHAPTER 13: LOFT FEATURES

- Rail Lofts
- Center Line Lofts
- Advanced Loft Options
- Chapter exercise

CHAPTER 14: DUPLICATION TOOLS

- Rectangular Feature Patterns
- Circular Feature Patterns
- Mirror Parts or Features
- Manipulate Patterns and Mirror Features
- Chapter exercise

CHAPTER 15: FEATURE RELATIONSHIPS

- Establishing Relationships
 - Controlling Relationships
 - Investigating Relationships
 - Changing Relationships
 - Chapter exercise
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CHAPTER 16: ASSEMBLY ENVIRONMENT

- Assembling Components using Constraints
- Content Center
- Assembly Browser
- Saving Files
- Chapter exercise

CHAPTER 17: JOINT CONNECTIONS

- Assembling Components using Joints
- Chapter exercise

CHAPTER 18: MANIPULATING ASSEMBLY DISPLAY

- Moving and Rotating Assembly Components
- Suppressing Constraints
- Component Display
- Selection Options in Assemblies
- Chapter exercise

CHAPTER 19: MODEL INFORMATION

- Measurement Tools
- Model Properties
- Chapter exercise

CHAPTER 20: DESIGN PRESENTATION AND ANIMATION

- Exploded View Presentations
- Chapter exercise

DAY 3

CHAPTER 21: ASSEMBLY TOOLS

- Replacing Components
- Restructuring Components
- Driving Constraints
- Contact Solver
- Interference
- Error Recovery
- Chapter exercise

CHAPTER 22: ASSEMBLY PARTS AND FEATURES

- Assembly Parts
- Assembly Features
- Chapter exercise

CHAPTER 23: ASSEMBLY BILL OF MATERIALS

- Create Virtual Components
- Create Bill of Materials
- Chapter exercise

CHAPTER 24: WORKING WITH PROJECTS

- New Projects
- Resolving Links
- The Vault Browser
- Chapter exercise

CHAPTER 25: DRAWING BASICS

- New Drawing Views
- Manipulating Views
- Chapter exercise

CHAPTER 26: DETAILING DRAWINGS

- Dimensions
- Drawing Sheets
- Parts List
- Balloons
- Styles and Standards
- Hatching
- Chapter exercise

CHAPTER 27: DRAWING ANNOTATIONS

- Text
- Symbols
- Hole and Thread Notes
- Chamfer Notes
- Center Marks and Center Lines
- Hole Tables
- Revision Tables and Tags
- Chapter exercise

CHAPTER 28: CUSTOMIZING AUTODESK INVENTOR

- Application Options

- Document Settings
 - File Properties
 - Changing Part Units
 - Command Customization
 - Chapter exercise
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