

# INVENTOR

## INTRODUCTION TO SOLID MODELING

### OVERVIEW

This course instructs users in best usage approaches for parametric design philosophy through a hands-on, practice-intensive curriculum. Users acquire the knowledge needed to complete the process of designing models from conceptual sketching, through to solid modeling, assembly design, and drawing production.

Not all aspects of Inventor can be addressed within the time limits of an introductory course. Advanced techniques and features are covered in more advanced level training classes.

### SPECIFICS

**Duration:** 5 Days

**Hours:** 9 am - 4 pm (including lunch hour)

**Cost:** \$1,895/person (includes training materials)

**Prerequisites:** As an introductory course, Introduction to Solid Modeling does not assume prior knowledge of any 3D modeling or CAD software.

Students do need to be experienced with the Windows operating system and a background in drafting of 3D parts is recommended.

\* This course is based on Autodesk® Official Training Guide (AOTG).

### TOPICS COVERED

#### • Introduction to Inventor

- Solid Modeling
- Inventor Fundamentals
- The Inventor Interface
- Model Manipulation

#### Creating the Base Feature

- Solid Base Features
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#### Sketching Geometry

- Sketch Geometry

#### Additional Sketching Tools

- Advanced Editing Tools
- Using Existing Geometry
- Over-Dimensioned Sketches
- Sketch Preferences

#### Sketched Secondary Features

- Extruded Secondary Features
- Revolved Secondary Features
- Editing Sketched Secondary Features
- 3D Grip Modification

#### 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

#### Work Features

- Work Planes
- Work Axes
- Work Points

#### Equations

- Equations
- Parameters

#### Additional Features and Functions

- Face Draft
- Split a Face or Part
- Shells
- Ribs
- Bend Part
- Reordering Features
- Inserting Features
- Suppressing Features

#### Fixing Problems

- Sketch Failure
- Feature Failure

#### Sweep Features

- Sweep Features

#### Loft Features

- Rail Lofts
- Center Line Lofts
- Advanced Loft Options

# INVENTOR

## INTRODUCTION TO SOLID MODELING (CONTINUED)

### TOPICS COVERED (CONTINUED)

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#### Duplication Tools

- Rectangular Sketch Patterns
- Circular Sketch Patterns
- Rectangular Feature Patterns
- Circular Feature Patterns
- Mirror Parts or Features
- Manipulate Patterns and Mirror Features

#### Feature Relationships

- Establishing Relationships
- Controlling Relationships
- Investigating Relationships
- Changing Relationships

#### Assembly Environment

- Assembling Components
- Content Center
- Assembly Browser
- Saving Files

#### Manipulating Assembly Display

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

#### Model Information

- Measurement Tools
- Model Properties
- Changing Part Units

#### Design Presentation and Animation

- Exploded View Presentations
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#### Assembly Tools

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

#### Assembly Parts and Features

- Assembly Parts
- Assembly Features

#### Working With Projects

- New Projects
- Resolving Links
- The Project Browser

#### Drawing Basics

- New Drawing Views
- Manipulating Views

#### Detailing Drawings

- Dimensions
- Parts List
- Balloons
- Style and Standards

#### Assembly Bill of Materials

- Create Virtual Components
- Create Bill of Materials

#### Drawing Annotations

- Text
- Symbols
- Hole and Thread Notes
- Chamfer Notes
- Center Marks and Centerlines
- Hole Tables
- Revision Tables and Tags

#### Customizing Inventor

- Application Options
- Document Settings
- File Properties
- Productivity Tips

#### Effective Modeling

- Design Considerations
- Modeling Tips and Techniques
- Model Investigation

#### DWG True Connect

- Introduction
- Inventor & AutoCAD Files
- Geometry Formatting Tools