



Autodesk Inventor Fundamentals (IGTAI)

Course Length: 5 days

Inventor Fundamentals introduces the concepts and techniques of 3D modeling with Autodesk Inventor. As an introductory class, it does not assume prior knowledge of any 3D modeling or CAD software. The objective of the Autodesk Inventor Fundamentals course is to teach students the basic part and assembly modeling techniques. Students will explore topics such as; the Autodesk Inventor interface, sketching tools, part modeling tools, assembly modeling tools, the Design Assistant, creation of drawing views, working drawing, and creating bills of materials. The structure of the course follows typical stages of using Inventor to create and edit parts, add them to assemblies, and generate drawings.

Prerequisites:

- A working knowledge of NT, 2000 or XP.
- A working knowledge of basic drafting procedures and terminology common to the mechanical design industry.

Day 1

Module 1: Introduction to Inventor

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

Module 2: Creating the Base Feature

- Grid and Axis Display
- Default Work Features
- Sketch Planes
- Sketching
- Dimensions
- Constraining

Module 3: Sketching Geometry

- Construction Geometry
- Slice Graphics
- Trim, Extend, Mirror, Fillets & Chamfers
- Assigning and Deleting Constraints

Module 4: Additional Sketch Tools

- Advanced Editing Tools
- Using Existing Geometry
- Resolving and Over-Dimensioned Sketch
- Setting Sketcher Preferences

Module 5: Creating Pick and Place Features

- Creating and Edge Chamfer
- Creating Constant Fillets
- Creating Variable Fillets
- Creating Face Fillets
- Creating Straight Holes
- Editing Pick and Place Features

Day 2

Module 6: Sketched Secondary Features

- Creating Extruded Secondary Features
- Creating Revolved Secondary Features
- Editing a Sketched Secondary Feature
- Creation Sequence

Module 7: Creating Work Features

- Creating Work Planes
- Creating Work Axes
- Creating Work Points

Module 8: Creating Relations

- Create Relations
- Add Parameters

Module 9: Additional Features and Functions

- Creating Threads
- Creating a Face Draft
- Creating Shells
- Creating Ribs
- Reordering Features
- Suppressing Features
- Changing Part Units

Module 10: Sweep Features Forms

- Creating Sweep Features

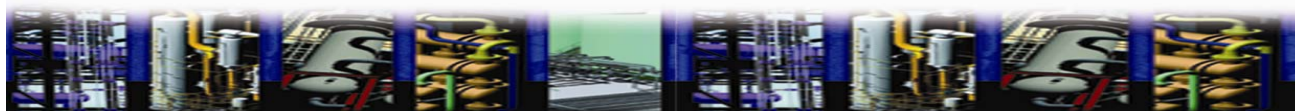
Module 11 Loft Feature Forms

- Creating Loft Features

Day 3

Module 12: Duplication Tools

- Creating Sketch Patterns
- Pattern Part Features
- Mirroring Parts or Part Features
- Manipulating Patterns and Mirror Features





Module 13: Feature Relationships

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

Module 14: Fixing Problems

- Sketch Failure
- Feature Failure

Module 15: Assembly Environment

- Assemble Components
- Assembly Examples
- Use the Content Center
- Assembly Browser
- Save Files

Module 16: Manipulating Assembly Display

- Move and Rotate Assembly Components
- Suppress Constraints
- Component Display
- Selection Options in Assemblies

Day 4

Module 17: Model Information

- Measurement Tools
- Model Properties
- Change Part Units

Module 18: Design Presentation and Animation

- Display Exploded Views
- Advanced Presentation File Tools

Module 19: Assembly Tools

- Replacing Components
- Restructuring Components
- Checking for Interference
- Error Recovery

Module 20: Working with Weldments

- Working with Weldments
- Create Filler Weld
- Create Cosmetic Weld
- Create Groove Weld

Module 21: Drawing Basics

- Creating a New Drawing & Drawing Views
- Manipulating Views

Module 22: Detailing Drawings

- Adding Dimensions

- Creating Part List

- Adding Balloons
- Defining Style and Standards

Day 5

Module 23: Drawing Annotations

- Create Text
- Add Hole and Thread Notes
- Add Chamfer Notes
- Centerlines and Center Marks

Module 24: Customizing Inventor

- Application Options
- Documentation Settings
- File Properties

Module 25: Effective Modeling

- Design Considerations
- Modeling Tips and Techniques
- Model Investigation
- Interactive Exercise
- Suggested Steps for the Interactive Exercise

Module 26: Working with Projects

- Create a New Project
- Resolve Links
- Manage the Project Browser
- Saving Files

Module 27: Creating Emboss and Decal Features

- Create Emboss Features
- Add Decals

