



# Autodesk® Inventor 11 Tutorial Series

## *Computer Based Training*

### ***What You Will Learn:***

Discover the program structure and essential features of Autodesk Inventor. Learn how to use Inventor's sketch tools, constraints, 3D space & work features, and how to create and maintain Inventor projects. You'll also learn how to create many advanced features, sheet metal and weldments.

### ***Topics covered on this DVD include:***

#### **Getting Started**

Getting started with Autodesk Inventor  
 Creating a Single User Project  
 File Types and Multiple Documents  
 Changing Inventor's Application Options  
 Inventor's User Interface and Help Tools  
 Inventor Command, customizing short cuts and Undo and Redo tool  
 Inventor's viewing tools

#### **Sketch, Constraining & Dimensioning**

Inventor's Sketch and Part Application Options, Units and Templates  
 Sketches and Default Planes, autoproject origin, planes, new sketch  
 Inventor's 2D sketching tools  
 Inventor's Precise Input tool  
 Selecting and Deleting Objects

#### **Sketch, Constraining & Dimensioning**

Measuring sketches and parts  
 2D sketch constraints  
 Construction geometry  
 Snaps and dragging  
 Applying sketch dimensions  
 Importing AutoCAD DWG file  
 Overview technique to open SAT, STEP, ProE, DXF, IGES files

#### **Creating & Editing Sketched Features**

Introduction to features  
 Extrude a sketch

### ***Prerequisites:***

Users should be familiar with Windows and the standard user interface tools such as the mouse and Windows Explorer.

### ***Requirements:***

Windows XP  
 DVD Drive  
 Pointing device  
 I Explorer 6+  
 Flash Plugin

**Creating & Editing  
Sketched Features**

Revolve a sketch  
Editing a feature and a features sketch  
How to use the the 3D Grip option and the Move Face tool.  
Rename feature, feature color, delete feature  
Sketched feature Options  
Define active sketch  
Project edges

**Creating and Editing  
Placed Features**

Create fillet features  
Create chamfer features  
Create hole features  
Create thread features  
Create shell features  
Create a face draft features  
Create work axis and work planes features  
Create work points and grounded work points  
Controlling the visibility of work features  
Creating circular and rectangular patterns  
Creating linear patterns

**Creating & Editing  
Drawing Views**

Starting and preparing a drawing  
Copy title blocks, borders or sketched symbols from one IDW file to another.  
Working with Styles  
Creating base and projected views  
Creating auxiliary, section and detail views  
Creating a broken views  
Creating a break out views  
Create draft views  
Editing drawing views  
Retrieving model dimensions and placing drawing dimensions  
Moving dimensions in a drawing view  
Adding centerlines to drawings

**Creating & Editing  
Drawing Views**

Placing text and symbols in a drawing

Placing hole notes and chamfer notes

Creating a hole table

Creating multiple sheets, copy and move views between sheets, create sheet format

Create baseline dimensions

Create ordinate dimension

Create drawing views with sketches

Create a table in a drawing

Plot multiple sheets

**Creating &  
Documenting  
Assemblies**

Creating an assembly file and the assembly browser

Inserting Parts and create parts in place

Create a subassembly and work with grounded parts.

Apply basic assembly constraints

Apply motion constraints

Assembly browser options

Enable, disable and control the visibility of components.

Edit components in an assembly and isolate constraint errors.

Create adaptive parts

Pattern assembly components

Analysis tools in an assembly

Drive assembly constraint to simulate motion

Create exploded view with a presentation file and create drawing views from them.

**Creating &  
Documenting  
Assemblies**

Create drawing views from an assembly and place balloons

Create a parts list and edit it via a bill of material

Create a revision table

**Advanced Sketching &  
Constraining  
Techniques**

Create ellipses

Import points and create splines

Create and edit pattern sketches.

Share and copy a sketch

Use the 2D mirror tool and symmetric

constraint

Slice the graphics in a part and assembly.

Sketch on another part's face.

**Advanced Sketching &  
Constraining  
Techniques**

Change the display of dimensions and create a relationship between dimensions.

Use parameters and link a spreadsheet.

**Advanced Part  
Modeling Techniques**

Use open profiles to create a feature

Create ribs and webs

Emboss text and sketches

Create sweep features

Create 3D sketches

Create a coil feature.

Create loft features

Create and edit surfaces

Split a face and part

Copy and paste features.

Mirror features on a part.

Suppress, Conditionally Suppress, and reorder features and feature rollback

Create a derived part and assembly.

Create AutoLimits

**Design Automation  
Techniques**

Create iMates.

Create, use and document iParts

Create, use and document iAssemblies

Create iFeatures and reuse them on other parts.

Create and work with design views.

Create flexible assemblies, positional representations and overlay drawing views.

Create level of detail in an assembly and use when creating drawing views.

Use the contact solver

Mirror components in an assembly

Copy components in an assembly

Create assembly features

2D Design layouts

Content Center

Design Accelerator

Frame Generator

**Sheet Metal Design**

Using sheet metal styles

Face tool

Contour Flange tool

Flange tool

HEM tool

Fold tool

Bend tool

CUT tool

CORNER SEAM tool

Corner Round & Chamfer tool

Punch Tool

Creating flat pattern

Common Sheet Metal & Feature Tools

Document sheet metal designs

**Weldments**

Creating a new weldment

Weld preparations

Create fillet welds

Creating Groove / Gap welds

Document weldments