

## **INVENTOR PARTS, ASSEMBLIES AND SHEET METAL**

### **3D mechanical parts and Sheet Metal design, visualization and documentation**

This course taught to create 3D parametric models with constrained geometry utilizing Autodesk Inventor. Topics include working with constrained geometry, creating and documenting assemblies, and advanced part modeling techniques.

Inventor has a dedicated Sheet Metal Design environment which includes a comprehensive set of tools to accelerate and simplify the design, documentation, and manufacture of complex sheet metal products.

#### **Course Objectives**

Start a part or model, understand sketch containment

Control 3D features and edit sketches

Generate 2D drawings from 3D models

Sheet Metal:

Model a parts assembly, manage a subassembly

Converting Solid Model to Sheet Metal

Flat Patterns

Sheet Metal Operations

Sheet Metal Cuts

Unfold and Refold

## **AUTODESK INVENTOR COURSE CONTENT**

### **Modeling**

2D Sketching Tools

3D Models from sketches

Work Planes

Extruded

Revolved

### **3D Modeling**

Profiles

Geometric constraints

Functions

Operations

Construction plans

Solid editing

Productivity features

### **Assembly**

Assembly constraints  
Geometric restrictions  
Degrees of freedom  
Mobility  
Assembly editing  
Correction for adjustment of assemblies  
Cinematic scenes

### **Presentations**

Dynamic expansive Isometrics  
Editing presentations  
Simultaneity  
Grouped and change of appearance of components  
Video recording

### **Drawings**

Views  
Cuts  
Details  
Scale and specifications of components and assemblies  
Preparation of lists of components  
Specification of drawing sheets

### **Sheet Metal**

Sheet Metal Environment  
Sheet Metal Styles  
Sheet Metal Operations  
Corner Rounds and Chamfers  
Corner Seams  
Sheet Metal Cuts  
Lofted Flanges and Rips  
Unfold and Refold  
Flat Pattern  
Sheet Metal Drawings  
Notes & Tables  
Converting Solid Model to Sheet Metal  
Non-Ruled Surfaces