



AUTODESK INVENTOR SHEET METAL TRAINING

Objectives

The main objective of this course is to teach users the advance commands and tools necessary for professional 3D design focusing on Sheet Metal using Inventor. After completing this course users will be able to:

- ✓ Understand the concept and techniques to design 3D model for Sheet Metal
- ✓ Create and edit sheet metal parts, generate flat patterns, and document the designs in drawings.

Pre-requisites

This guide is designed for an Inventor User. It is recommended that you have a working knowledge of:

- Microsoft® Windows® 7, Microsoft® Windows® 8 or Microsoft® Windows® 10
- Fundamental Inventor Part, Assembly, and Drawing
- A background in designing and drafting 3D parts is recommended.

Chapter 1: Introduction to Sheet Metal Modelling

- Sheet Metal Concepts
- Sheet Metal Terminology
- Sheet Metal Environment
- Sheet Metal Design Process

Chapter 2: Sheet Metal Base Features

- Applying Existing Sheet Metal Defaults
- Creating a Face as a Base Feature
- Creating a Contour Flange as a Base Feature
- Creating a Contour Roll as a Base Feature

Chapter 3: Sheet Metal Secondary Features

- Sheet Metal Parameters
- Bend Relief Shapes
- Faces as Secondary Features
- Contour Flanges as Secondary Features
- Contour Rolls as Secondary Features

Chapter 5: Bending Sheet Metal

- Hems
- Folds
- Bends

Chapter 6: Corner Rounds and Chamfers

- Creating Corner Rounds
- Creating Corner Chamfers

Chapter 4: Flanges

- Creating Flanges
- Edge Flanges
- Loop Flanges
- Corner Relief Options
- Corner Relief Shapes for 2 Bend Intersections
- Corner Relief Size
- Corner Mitering

Chapter 7: Sheet Metal Cuts

- Creating Cut Features
- Creating Straight Holes
- Using Punch Tool Features
- Creating a Punch Tool
- Cuts Using Surfaces

Chapter 8: Corner Seams

- Creating Corner Seams and Miters
- Creating Corner Rips
- Converting Corner Seams and Bends

Chapter 9: Flat Pattern Environment

- Creating Flat Patterns
- Orienting Flat Patterns
- Punch Representations
- Bend Angle
- Flat Pattern Cleanup
- Exporting to DXF/DWG

Chapter 13: Documentation and Annotation

- Sheet Metal Drawing Terminology
- Creating Sheet Metal Drawings
- Bend & Punch Notes
- Bend Tables
- Punch Tables
- Bend Order
- Cosmetic Centerlines

Chapter 10: Lofted Flange and Rips

- Lofted Flange
- Rip

Chapter 11: Unfold and Refold

- Unfold and Refold

Chapter 12: Multi-Body Sheet Metal Modeling

- Multi-Body Modeling
- Creating the First Solid Body
- Creating Additional Solid Bodies
- Assigning Features to Solid Bodies
- Manipulating Solid Bodies
- Solid Body Display
- Solid Body Properties

Chapter 14: Converting Parts to Sheet Metal

- Converting Solid Models to Sheet Metal
- Common Shapes to Convert to Sheet Metal
- Non-Ruled Surfaces