

Autodesk Inventor Sheet Metal Design Training

Course Length: 2 days

The Autodesk Inventor Sheet Metal Design training course introduces the concepts and techniques of sheet metal modelling with the Autodesk Inventor software.

The structure of the course follows the typical stages of using the Autodesk Inventor software. That is, to create and edit sheet metal parts, generate flat patterns, and document the designs in drawings.

Topics Covered

- Autodesk Inventor Sheet Metal interface
- Sheet metal design process
- Creating base Faces, Contour Flanges, and Contour Rolls
- Creating secondary Faces, Contour Flanges, and Contour Rolls
- Sheet metal parameters
- Creating Flanges
- Creating Hems, Folds, and Bends
- Corner Rounds and Chamfers
- Sheet Metal Cuts (Holes, Cuts, and Punch Features)
- Corner Seams (Seams and Mitres)
- Generating Flat Patterns
- Lofted Flanges
- Rips
- Unfolding and Refolding
- Multi-Body Sheet Metal Modelling
- Documentation and Annotation of drawings
- Converting solid models to sheet metal models
- Sheet Metal Styles

Prerequisites

The material covered in this training course assumes a mastery of Autodesk Inventor basics as taught in Autodesk Inventor: Introduction to Solid Modelling.

Knowledge of sheet metal processing is an asset, but not required.

Training Guide Contents

Chapter 1: Introduction to Sheet Metal Modelling

- 1.1 Sheet Metal Concepts
- 1.2 Sheet Metal Terminology
- 1.3 Sheet Metal Environment
- 1.4 Sheet Metal Design Process

Chapter 2: Sheet Metal Base Features

- 2.1 Applying Existing Sheet Metal Defaults
- 2.2 Creating a Face as a Base Feature
- 2.3 Creating a Contour Flange as a Base Feature
- 2.4 Creating a Contour Roll as a Base Feature

Chapter 3: Sheet Metal Secondary Features

- 3.1 Sheet Metal Parameters
- 3.2 Bend Relief Shapes
- 3.3 Faces as Secondary Features
- 3.4 Contour Flanges as Secondary Features
- 3.5 Contour Rolls as Secondary Features

Chapter 4: Flanges

- 4.1 Creating Flanges
- 4.2 Corner Relief Options

Chapter 5: Bending Sheet Metal

- 5.1 Hems
- 5.2 Folds
- 5.3 Bends

Chapter 6: Corner Rounds and Chamfers

- 6.1 Creating Corner Rounds
- 6.2 Creating Corner Chamfers

Chapter 7: Sheet Metal Cuts

- 7.1 Creating Cut Features
- 7.2 Creating Straight Holes
- 7.3 Using Punch Tool Features
- 7.4 Creating a Punch Tool
- 7.5 Cuts Using Surfaces

Chapter 8: Corner Seams

- 8.1 Creating Corner Seams and Mitres
- 8.2 Creating Corner Rips
- 8.3 Converting Corner Seams and Bends

Chapter 9: Flat Pattern Environment

- 9.1 Creating Flat Patterns
- 9.2 Orienting Flat Patterns
- 9.3 Punch Representations
- 9.4 Bend Angle
- 9.5 Flat Pattern Clean-up
- 9.6 Exporting to DXF/DWG

Chapter 10: Lofted Flange and Rips

- 10.1 Lofted Flange
- 10.2 Rip

Chapter 11: Unfold and Refold

- 11.1 Unfold and Refold

Chapter 12: Multi-Body Sheet Metal Modelling

- 12.1 Multi-Body Modelling

Chapter 13: Documentation and Annotation

- 13.1 Sheet Metal Drawing Terminology
- 13.2 Creating Sheet Metal Drawings
- 13.3 Bend & Punch Notes
- 13.4 Bend Tables
- 13.5 Punch Tables
- 13.6 Bend Order
- 13.7 Cosmetic Centrelines

Chapter 14: Converting Parts to Sheet Metal

- 14.1 Converting Solid Models to Sheet Metal
- 14.2 Non-Ruled Surfaces

Appendix A: Sheet Metal Rules

- A.1 Working with Sheet Metal Rules
- A.2 Sheet, Bend, and Corner Tab Options
- A.3 Bend Tables

Appendix B: Additional Practices

Appendix C: Autodesk Inventor Certification Exam Objectives