

Autodesk Inventor Sheet Metal Training Course

Course Overview

Course Length: 8-Hours

- 8-Hours - 4 x 2-hour training sessions
- Remote training over MS Teams
- Sessions are recorded and download links for each session are provide for future use
- Training for up to 1-3 People
- Sessions can start within 7 days upon ordering
- Flexible sessions (i.e. consecutive or Monday, Wednesday, Friday, etc.)
- Courses are private and topics can be customised to suit
- Includes Certificate of Completion

This Autodesk Inventor Sheet Metal Design course introduces the concepts and techniques used to create, edit, and document sheet metal components in Autodesk Inventor. The course follows the typical sheet metal workflow, from creating base features and applying design rules through to generating flat patterns and production-ready drawings, giving participants practical skills they can apply in real manufacturing and design environments.

Topics Covered

This course covers the essential tools, workflows, and best practices required to model sheet metal parts efficiently in Autodesk Inventor, prepare them for manufacture, and communicate designs clearly through drawings.

- Autodesk Inventor Sheet Metal interface and design workflow
- Creating base faces, contour flanges, and contour rolls
- Creating secondary features, flanges, hems, folds, and bends
- Working with sheet metal parameters, corner reliefs, seams, and rips
- Creating cuts, holes, punch features, and corner treatments
- Generating, orienting, and cleaning up flat patterns for manufacture
- Documenting sheet metal parts with drawings, bend notes, and annotations
- Converting solid models to sheet metal models and working with styles

Prerequisites

Participants should have a solid understanding of Autodesk Inventor basics, equivalent to Autodesk Inventor Introduction to Solid Modelling or similar experience. A general familiarity with 3D part modelling, sketching, and the Windows operating system is recommended.

Knowledge of sheet metal manufacturing processes is helpful, but not essential. The course is designed for current Inventor users who want to build practical skills in creating flat patterns, applying sheet metal features, and preparing models for documentation and manufacture.

Training Guide Contents

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 4: Flanges

- Creating Flanges
- Corner Relief Options

Chapter 5: Bending Sheet Metal

- Hems
- Folds
- Bends

Chapter 6: Corner Rounds and Chamfers

- Creating Corner Rounds
- Creating Corner Chamfers

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 Mitres
- 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 Clean-up
- Exporting to DXF/DWG

Chapter 10: Lofted Flange and Rips

- Lofted Flange
- Rip

Chapter 11: Unfold and Refold

- Unfold and Refold

Chapter 12: Multi-Body Sheet Metal Modelling

- Multi-Body Modelling

Chapter 13: Documentation and Annotation

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

Chapter 14: Converting Parts to Sheet Metal

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

Appendix A: Sheet Metal Rules

- Working with Sheet Metal Rules
- Sheet, Bend, and Corner Tab Options
- Bend Tables