

Autodesk® Inventor® – Sheet Metal Design

Brief Synopsis of Class Contents:

Autodesk® Inventor® 3D CAD software offers an easy-to-use set of tools for 3D mechanical design, documentation, and product simulation. In this course, students are introduced to the concepts and techniques of sheet metal modeling with the Autodesk® Inventor® software .

Learning Objectives:

- 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 Miters)
- Generating Flat Patterns
- Lofted Flanges
- Rips
- Unfolding and Refolding
- Documentation and Annotation of drawings
- Converting solid models to sheet metal models
- Sheet Metal Styles

Courseware:

Ascent Autodesk® Inventor® Sheet Metal Design

Number of Days:

4 Half Day Sessions

Continuing Education Hours:

16 hours

Who Should Attend:

This courseware is designed for current Inventor users who are ready to utilize the sheet metal capabilities with the Autodesk® Inventor® software.

Prerequisites:

The topics presented in this course assume prior knowledge of 3D solid part modeling using the Autodesk Inventor software. Students also need to be experienced with the Windows operating system. In addition, some background with designing and drafting 3D parts is recommended. Although it is not required, knowledge of sheet metal processing is helpful as students learn to use the sheet metal design tools

System and Software Requirements:

<http://www.asti.com/LiveLab-Learning-amp-Training/LiveLab-System-Requirements>

FAQs and Cancellation Policy:

<http://www.asti.com/LiveLab-Learning-amp-Training/LiveLab-FAQS>

Class Outline and Topics:

Chapter 1: Introduction to Sheet Metal Modeling

- 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
- 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 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

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 14: Converting Parts to Sheet Metal

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

