

SOLIDWORKS SHEET METAL

Pre-Requisites: SOLIDWORKS Essentials

Daily Schedule: 8:30 a.m. - 4:30 p.m.

Length: 2 Days

This comprehensive course covers all aspects of sheet metal parts in SOLIDWORKS, from creating a base flange to process plans and flat blank drawings. Learn the difference between converting a part to sheet metal and truly designing a sheet metal part.

New techniques and features like tabs, edge flanges, and miter flanges make SOLIDWORKS an even better sheet metal design tool than before. Even if you've been trained in earlier versions of SOLIDWORKS sheet metal, taking this one-day course will be well worth your time!

Introduction

- » About This Course
- » Windows® XP
- » Use of Color

Lesson 1: Modeling Sheet Metal Parts

- » What are Sheet Metal Parts?
- » Sheet Metal Methods
- » Base Flange
- » Flat Pattern
- » Edge Flanges
- » Editing Sheet Metal Settings
- » Cuts in Sheet Metal
- » Break Corner
- » Sheet Metal Parts in Drawings

Lesson 2: Sheet Metal Convert Method

- » Sheet Metal Conversion Topics
- » Converting to Sheet Metal
- » Imported Geometry to Sheet Metal
- » Using the Rip Feature
- » Adding Bends in Place of Sharp Corners
- » Sheet Metal Features
- » Making Changes
- » Adding a Welded Corner

Lesson 3: Multibody Sheet Metal Parts

- » Multibody Sheet Metal Parts
- » Methods to Create Multibody Sheet Metal Parts
- » Creating Multibodies by Sketching
- » Miter Flange
- » The Cut List Folder
- » Cuts using Multibodies
- » Patterning Sheet Metal Bodies
- » Sheet Metal Properties
- » Multibody Drawings
- » Using Mirror and Insert Part
- » Interfering Bodies
- » Exporting Sheet Metal Bodies
- » Using Split

Lesson 4: Sheet Metal Forming Process

- » Sheet Metal Forming Tools
- » Modifying an Existing Forming Tool
- » Creating a Custom Forming Tool

Lesson 5: Additional Sheet Metal Features & Techniques

- » Additional Sheet Metal Features
- » Using Symmetry
- » Additional Modeling Techniques
- » In-Context Methods
- » Process Plans

