3S SOLIDWORKS   invenio

Training
SolidWorks Sheet Metal

Description	Sheet Metal teaches you how to build sheet metal parts using SOLIDWORKS mechanical design automation software. Building standalone sheet metal parts, and converting conventional parts to sheet metal, including in assembly context, are covered.
Prerequisites	SolidWorks Essentials course
Duration	2 Days

Course Outline

Introduction	About This Course
Lesson 1	**Basic Flange Features** • What are Sheet Metal Parts? • Sheet Metal Methods • Unique Sheet Metal Items • Flange Method • Base Flange/Tab • Sheet Metal Parameters • Sheet Metal Bend Features • Flat-Pattern Feature • Additional Flange Features • Editing the Flange Profile • Edge and Miter Flanges • Hem Feature • Tab Features • Cuts in Sheet Metal
Lesson 2	**Working with the Flat Pattern** • Working with the Flat Pattern • Flat Pattern Settings • Features for Manufacture • Corner-Trim Feature • Corners in the Formed State • Closed Corner and Corner Relief • Break Corner/Corner Trim • Producing the Flat Pattern • Sheet Metal Cut List Properties • Flat Pattern Drawing Views and Properties • Drawing Document Properties • Sheet Metal Tables • Cut List Properties as a Note • Exporting the Flat Pattern

Lesson 3	**Additional Sheet Metal Techniques**
	• Additional Sheet Metal Methods
	• Designing from the Flat
	• Sketched Bend Feature
	• Jog Feature
	• Adding Features in an Unfolded State
	• Unfold and Fold
	• Swept Flange and Flat Pattern options
	• Lofted Bends
	• Lofted bends in the Design Library
Lesson 4	**Converting to Sheet Metal**
	• Sheet Metal Conversion
	• Insert Bends Method
	• Imported Geometry to Sheet Metal
	• Adding Rips
	• Insert Bends
	• Making Changes
	• Welded Corner
	• Converting Cones and Cylinders
	• Convert to Sheet Metal
Lesson 5	**Multibody Sheet Metal Parts**
	• Multibodies with Base Flange
	• Sheet Metal Parameters for Multibodies
	• Cut List Item Properties for Multibodies
	• Flat Pattern Drawing Views for Multibodies
	• Cut List Balloon Annotations
	• Exporting to DXF/DWGs with Multibodies
	• Convert with Multibodies
	• Hiding and Showing Bodies
	• Using Split with Sheet Metal Parts
	• Patterning for Multibodies
	• Using Edge Flanges to Merge Bodies
	• Interfering Bodies
	• Combining Sheet Metal with Other Bodies
Lesson 6	**Forming Tools and Gussets**
	• Sheet Metal Forming Tools
	• Standard Forming Tools
	• Form Tool Features in the Flat
	• Part Document Properties
	• Custom Forming Tools
	• Split Line
	• Forming Tool
	• Form Tools in Drawings
	• Sheet Metal Gusset
Lesson 7	**Additional Sheet Metal Functions**
	• Cross-Breaks
	• Vent Features
	• Mirror Part
	• Tab and Slot
	• Process Plans and Costing

Training
SolidWorks Weldments

Description

Weldments teaches you how to create welded structures with standard structural members. Weld beads are also covered.

Prerequisites

SolidWorks Essentials course

Duration

1 Day

Course Outline

Introduction	About This Course
Lesson 1	**Weldment Features** • Weldments • Structural Members • Groups vs Structural Members • Adding Plates and Holes • Gussets and End Caps • Using Symmetry • Advantages and Limitations of a Multibody Part
Lesson 2	**Working with Weldments** • Managing the Cut List • Cut List Item Names • Accessing Properties • Cut-List Properties Dialogue • Structural Member Properties • Adding Cut List Properties • Bounding Boxes in Weldments • Generating Cut List Items • Custom Structural Member Profiles • Defining Material • Creating Custom Profiles • Standard or Configured Profiles • Inserting Existing Parts • When to Use an Assembly
Lesson 3	**Configuring and Detailing Weldments** • Weldment Configurations • Post-Assembly Machining Features • Weldment Drawings • Drawing Views of Individual Bodies • Representing Welds
Lesson 4	**Working with Bent Structural Members** • Working with Bent Structural Members • 3D Sketching