

Course Outline

SOLIDWORKS Sheetmetal	
Description	This course teaches you how to build sheet metal parts using SOLIDWORKS. Building standalone sheet metal parts, and converting conventional parts to sheet metal, including in assembly context, are covered.
Prerequisites	Equivalent or in addition to SOLIDWORKS Essentials
Duration	2 days
Delivery Mode	Face to Face OR Online

<p>Introduction About This Course</p> <p>Lesson 1: Sheet Metal Flange Method 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</p> <p>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</p>	<p>Lesson 3: Multibody Sheet Metal Parts Multibody Sheet Metal Parts Methods to Create Multibody Sheet Metal Parts Creating Multibodies by Sketching Mitre 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</p> <p>Lesson 4: Sheet Metal Forming Tools Sheet Metal Forming Tools Modifying an Existing Forming Tool Creating a Custom Forming Tool</p>	<p>Lesson 5: Additional Sheet Metal Features and Techniques Additional Sheet Metal Features Using Symmetry Additional Modelling Techniques In-Context Methods Process Plans</p>
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