



SolidWorks Essentials 2012 Training Course

Duration: 4 days • Prerequisites: Mechanical design & Windows™ operating system experience. Review the SolidWorks tutorials and getting started guide included with your software.

Course Outline

- LESSON 1** SolidWorks Basics & the User Interface
- LESSON 2** Introduction to Sketching
- LESSON 3** Basic Part Modelling
- LESSON 4** Modelling a Casting or Forging
- LESSON 5** Patterning
- LESSON 6** Revolved Features
- LESSON 7** Shelling and Ribs
- LESSON 8** Editing: Repairs
- LESSON 9** Editing: Design Changes
- LESSON 10** Configurations
- LESSON 11** Using Drawings
- LESSON 12** Bottom-Up Assembly Modelling
- LESSON 13** Using Assemblies



Training Content

LESSON 1

SolidWorks Basics and the User Interface

- What is the SolidWorks Software?
- Design Intent
- File References
- Opening Files
- The SolidWorks User Interface
- Using the Command Manager

LESSON 2

Introduction to Sketching

- 2D Sketching
- Stages in the Process
- Saving Files
- What are We Going to Sketch?
- Sketching
- Sketch Entities
- Basic Sketching
- Rules That Govern Sketches
- Design Intent
- Sketch Relations
- Dimensions
- Extrude
- Sketching Guidelines

LESSON 3

Basic Part Modelling

- Basic Modeling
- Terminology
- Choosing the Best Profile
- Choosing the Sketch Plane
- Details of the Part
- Boss Feature
- Sketching on a Planar Face
- Cut Feature
- Using the Hole Wizard
- View Options
- Filletting
- Detailing Basics
- Drawing Views
- Center Marks
- Dimensioning
- Changing Parameters

LESSON 4

Modelling a Casting or Forging

- Case Study: Ratchet
- Design Intent
- Boss Feature with Draft
- Symmetry in the Sketch
- Sketching Inside the Model
- View Options
- Using Model Edges in a Sketch
- Creating Trimmed Sketch Geometry

LESSON 5

Patterning

- Why Use Patterns?
- Reference Geometry
- Linear Pattern
- Circular Patterns
- Mirror Patterns
- Using Pattern Seed Only
- Sketch Driven Patterns

LESSON 6

Revolved Features

- Case Study: Handwheel
- Design Intent
- Revolved Features
- Building the Rim
- Building the Spoke
- Edit Material
- Mass Properties
- File Properties
- SolidWorks SimulationXpress
- Using SolidWorks SimulationXpress
- The SimulationXpress Interface

LESSON 7

Shelling and Ribs

- Shelling and Ribs
- Analyzing and Adding Draft
- Other Options for Draft
- Shelling
- Planes
- Ribs
- Full Round Fillets
- Thin Features

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 **SOLIDWORKS**
LET'S GO DESIGN



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LESSON 8

Editing: Repairs

- Part Editing
- Editing Topics
- Sketch Issues
- Freezing Features
- FilletXpert
- DraftXpert

LESSON 9

Editing: Design Changes

- Part Editing
- Design Changes
- Information From a Model
- Rebuilding Tools
- Sketch Contours

LESSON 10

Configurations

- Configurations
- Creating Configurations
- Using Configure
- Dimension/Feature
- Using Global Variables, Equations
- Global Variables
- Equations
- Modeling Strategies for Configurations
- Editing Parts that Have Configurations
- Design Library
- In the Advanced Course

LESSON 11

Using Drawings

- More About Making Drawings
- Section View
- Model Views
- Broken View
- Detail Views
- Drawing Sheets and Sheet Formats
- Projected Views
- Annotations

LESSON 12

Bottom-Up Assembly Modelling

- Case Study: Universal Joint
- Bottom-Up Assembly
- Creating a New Assembly
- Position of the First Component
- FeatureManager Design Tree and Symbols
- Adding Components
- Using Part Configurations in Assemblies
- Subassemblies
- Smart Mates
- Inserting Subassemblies
- Pack and Go

LESSON 13

Using Assemblies

- Using Assemblies
- Analyzing the Assembly
- Checking for Clearances
- Changing the Values of Dimensions
- Exploded Assemblies
- Explode Line Sketch
- Bill of Materials
- Assembly Drawings

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