



SOLIDWORKS Essentials

Length: 5 days

Prerequisite: Mechanical design experience and experience with the Windows operating system.

Description: SOLIDWORKS Essentials teaches you how to use SOLIDWORKS mechanical design automation software to build parametric models of parts and assemblies, and how to make drawings of those parts and assemblies.

The topics covered in this course are:

Introduction

About This Course

Windows

Use of Color

More SOLIDWORKS Training Resources

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 Modeling

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

Lesson 4: Symmetry and Draft

- 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
- Copy and Paste Features

Lesson 5: Patterning

- Why Use Patterns?
- Linear Pattern
- Circular Patterns
- Reference Geometry
- Planes
- Mirror Patterns
- Using Pattern Seed Only
- Up To Reference
- 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
Ribs
Full Round Fillets
Thin Features

Lesson 8: Editing: Repairs

Part Editing
Editing Topics
Sketch Issues

Lesson 9: Editing: Design Changes

Part Editing
Design Changes
Information from a Model
Rebuilding Tools
Sketch Contours
Replace Sketch Entity

Lesson 10: Configurations

Configurations
Using Configurations
Other Methods to Create Configurations
Modeling Strategies for Configurations
Editing Parts that Have Configurations



Design Library
In the Advanced Course...

Lesson 11: Global Variables and Equations

Using Global Variables and Equations
Renaming Features and Dimensions
Design Rules Using Global Variables and Equations
Global Variables
Equations
Using Operators and Functions

Lesson 12: Using Drawings

More about Making Drawings
Section View
Model Views
Broken View
Detail Views
Drawing Sheets and Sheet Formats
Annotations

Lesson 13: Bottom-Up Assembly Modeling

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

Lesson 14: 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

Appendix A: Templates

Options Settings
Document Templates