

## Training Agenda

SOLIDWORKS Essentials is a four-day training class that teaches how-to use SOLIDWORKS to build 3D parametric models of parts and assemblies and prepare drawings of those parts and assemblies.

The SOLIDWORKS Essentials training consists of the following lessons:

### Lesson 1: SOLIDWORKS Basics

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

### Lesson 2: Introduction to Sketching

- 2D Sketching
- Stages in the Process
- Saving Files
- Sketching and Sketch Entities
- Basic Sketching
- Rules That Govern Sketches
- Design Intent
- Sketch Relations
- Dimensions
- Extrude
- Sketching Guidelines

### Lesson 3: Basic Part Modeling

- Basic Modeling and 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: Modeling 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
- Copy and Paste Features

### Lesson 5: Patterning

- Why Use Patterns?
- Linear Pattern
- Circular Patterns
- Reference Geometry
- Planes
- 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

### Lesson 7: Shelling and Ribs

- Shelling and Ribs
- Analyzing and Adding Draft
- Other Options for Draft
- Shelling
- Planes

# SOLIDWORKS Essentials

- ▣ Ribs
- ▣ Full Round Fillets
- ▣ Thin Features

## 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
- ▣ Replace Sketch Entity

## Lesson 10: Configurations

- ▣ Using Configurations
- ▣ Modeling Strategies for Configurations
- ▣ Editing Parts that Have Configurations
- ▣ Design Library

## Lesson 11: Global Variables and Equations

- ▣ Using Global Variables and Equations
- ▣ Renaming Features and Dimensions
- ▣ Design Rules Using Global Variables
- ▣ 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

## Format

Classes are scheduled at a GoEngineer training facility and taught by certified SOLIDWORKS instructors. Training manuals will be provided to you on the first day of class. This training may also be taught as a custom class at your location for an additional cost.

## Prerequisites

We recommend completing the introductory SOLIDWORKS Tutorials, lessons one through three.