



SolidWorks Essentials Course Details 1/2

1: SolidWorks Basics User Interface

What is the SolidWorks Software
Design Intent
File References
Opening Files
The SolidWorks User Interface

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

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
Using the Hole Wizard
View Options
Filleting
Detailing Basics
Drawing Views
Center Marks
Dimensioning
Changing Parameters

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
Using Copy and Paste
Editing Features

5: Patterning

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

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

7: Shelling and Ribs

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

8: Editing: Repairs

Part Editing
Editing Topics
Sketch Issues
FilletXpert
DraftXpert

9: Editing: Design Changes

Part Editing
Design Changes
Information From a Model
Rebuilding Tools

Sketch Contours
Editing with Instant 3D

10: Configurations of Parts

Configurations
Using Configurations
Creating Configurations
Editing Parts that Have Configurations
Design Library

11: Design Tables and Equations

Design Tables
Link Values
Equations
Design Tables
Existing Design Tables
Modeling Strategies for Configurations
In the Advanced Course...

12: 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
Using Part Configurations in Assemblies
Sub-assemblies
Smart Mates
Inserting Sub-assemblies
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

continued....



SolidWorks Essentials Course Details 2/2



14: Drawing Sheets and Views

Drawing Sheets and Views
Terminology
Drawing Views
Sketching in Drawing Views
View Settings
Centermarks and Centerlines
Model Edges in the View

15: Dimensions

Dimensions
Moving and Deleting Dimensions
Dimension Properties

16: Annotations

Adding Annotations
Annotation Types
Blocks

17: Sheet Formats and Templates

Sheet Formats and Templates
Drawing Templates
Properties in the Template
User Defined Properties
Customizing a Sheet Format
Importing Legacy Data
Define Title Block
Updating Sheet Formats

18: Assembly Drawing Views

Assembly Drawing Views
Creating Views of Assemblies

19: Bill of Materials and Tables

Creating and Managing a BOM
The Bill of Materials
Adding a BOM
Modifying the BOM
Tabulated Bill of Materials
Design Tables in the Drawing

20: Performance and Display Issues

Performance and Display Issues
Large Assembly Mode
Lightweight Drawings
Detached Drawings
Display Issues in Drawing Views

21: Drawing References and Comparison

Reusing a Drawing File
Changing Drawing References
Using DrawCompare
Design Checker