

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.

Course Syllabus

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
- Filletting
- 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





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CERTIFIED TRAINING CENTER**

CADIMENSIONS TRAINING CATALOG

