



SOLIDWORKS ESSENTIALS

Pre-Requisites: Familiarity with Microsoft Windows

Daily Schedule: 8:30 a.m. - 4:30 p.m.

Length: 5 Days

This is an excellent introduction to SOLIDWORKS, giving the student the skills and knowledge necessary to make full use of SOLIDWORKS' power to model and annotate real products. Taught from the end-user's viewpoint, SOLIDWORKS will come alive as a real-world tool applicable to everyday engineering problems, as opposed to a static computer program.

The focus of this course is on designing parts, starting with proper sketching techniques through feature creation and editing to configurations and troubleshooting. Students are also introduced to assemblies and basic drawing sheet creation. This is a must-have course for anyone using SOLIDWORKS.

Graphics Systems Essentials classes now include the added value of File Management content. File Management teaches users how to manage SOLIDWORKS files. All content covered in our one-day class is now included in this Essentials course.

Introduction

- » About This Course
- » Windows® XP
- » Use of Color

Lesson 1: SOLIDWORKS Basics & User Interface

- » What is the SOLIDWORKS Software
- » Design Intent
- » File References
- » Opening Files
- » The SOLIDWORKS User Interface

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
- » Using the Hole Wizard
- » View Options
- » Filletting
- » Detailing Basics
- » Drawing Views
- » Center Marks
- » Dimensioning
- » Changing Parameters





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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
- » Using Copy and Paste

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 & 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
- » FilletXpert
- » DraftXpert

Lesson 9: Editing - Design Changes

- » Part Editing
- » Design Changes
- » Information From a Model
- » Rebuilding Tools
- » Sketch Contours
- » Editing with Instant 3D

Lesson 10: Configurations

- » Configurations
- » Using Configurations
- » Creating Configurations
- » Link Values
- » Equations
- » Configure Dimension / Feature
- » Modeling Strategies for Configurations
- » Editing Parts that Have Configurations
- » Design Library

Lesson 11: Using Drawings

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



These courses are taught from the official course curriculum from SOLIDWORKS Corporation, with additional information from Graphics Systems instructors.

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- » Projected Views
- » Annotations

Lesson 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

Appendix:

- » Options Settings
- » Document Templates



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