SOLIDWORKS Essentials

Overview

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

Prerequisites

It is recommended that delegates have a working knowledge of one or more of the following:

- Attended a Introduction to Technical Drawing OR have experience with draughting, design, or engineering principles.
- Comper Literacy Skills
- Understanding and Reading of Technical Drawings

Available Exams and Certifications

General Certifications:

- CSWA (Certified SOLIDWORKS Associate)
- CSWP (Certified SOLIDWORKS Professional)
- CSWE (Certified SOLIDWORKS Expert)
- CPPA (Certified PDM Professional Administrator)
- CSDA (Certified Sustainable Design Associate)

Product Specific Certifications:

- CSWPA – Sheet Metal
- CSWPA – Weldments
- CSWPA – Surfacing
- CSWPA – Mold Tools
- CSWPA – Drawing Tools

Course Outline

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

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

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

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

Patterning

- Why Use Patterns?
- Linear Pattern
- Circular Patterns
- Reference Geometry
- Planes
- Mirror Patterns
- Using Pattern Seed Only
- Up To Reference
- Sketch Driven Patterns

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

Shelling and Ribs

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

- Full Round Fillets
- Thin Features

Editing: Repairs

- Part Editing
- Editing Topics
- Sketch Issues

Editing: Design Changes Part

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

Configurations

- Configurations
- Using Configurations
- Other Methods to Create Configurations
- Renaming Features and Dimensions
- Design Rules Using Global Variables and Equations
- Using Global Variables and Equations
- Equations
- Using Operators and Functions
- Modeling Strategies for Configurations
- Editing Parts that Have Configurations
- Design Library
- In the Advanced Course

Using Drawings

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

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

Using Assemblies

- Using Assemblies
- Analyzing the Assembly
- Checking for Clearances
- Changing the Values of Dimensions
- Exploded Assemblies
- Explode Line Sketch
- Bill of Materials