

# TRAINING COURSE PROSPECTUS

## **SOLIDWORKS Essentials**

**Duration:**

4 Days

**Prerequisites:**

Mechanical design experience; experience with the Windows™ operating system

**Description:**

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.

The topics covered in this course are:

### **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 Modelling**

- Basic Modelling
- 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
- Editing Tools

# TRAINING COURSE PROSPECTUS

- 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

## 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

- Design Intent
- Revolved Features
- Building the Rim
- Building the Spoke
- Edit Material
- Mass Properties
- File Properties

## 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

# TRAINING COURSE PROSPECTUS

- FilletXpert
- DraftXpert

## Lesson 9: Editing: Design Changes

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

## Lesson 10: Configurations

- Configurations
- Using Configure Dimension/Feature
- Other Methods to Create Configurations
- Using Global Variables and Equations
- Creating Equalities
- Global Variables
- Defining the Overall Width
- Creating a Minimum Edge Distance
- 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
- Projected Views
- Annotations

## Lesson 12: Bottom-Up Assembly Modelling

- Bottom-Up Assembly
- Creating a New Assembly
- Position of the First Component
- Feature Manager Design Tree and Symbols
- Adding Components
- Mating Components
- Using Part Configurations in Assemblies
- Sub-assemblies
- Smart Mates
- Inserting Subassemblies
- Pack and Go

## Lesson 13: Using Assemblies

# TRAINING COURSE PROSPECTUS

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



# TRAINING COURSE PROSPECTUS

## **SOLIDWORKS Drawings**

**Duration:**

2 Days

**Prerequisites:**

SOLIDWORKS Essentials

**Description:**

This course teaches you how to make drawings of SOLIDWORKS parts and assemblies.

The topics covered in this course are:

### **Lesson 1: Drawing Sheets and Views**

- Drawing Sheets and Views
- Sheets and Formats
- Terminology
- Drawing Views
- Sketching in Drawing Views
- View Settings
- center marks and Centerlines
- Model Edges in the View

### **Lesson 2: Dimensions**

- Dimensions
- Manipulating Dimensions
- Dimension Properties

### **Lesson 3: Annotations**

- Adding Annotations
- Annotation Types
- Blocks

### **Lesson 4: Assembly Drawing Views**

- Assembly Drawing Views
- Creating Views of Assemblies

### **Lesson 5: Sheet Formats and Templates**

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

### **Lesson 6: Bill of Materials and Tables**

- Creating and Managing a Bill of Materials
- The Bill of Materials
- Adding a BOM
- Modifying the BOM
- Tabulated Bill of Materials
- Balloons

# TRAINING COURSE PROSPECTUS

- Tables in the Drawing

## **Lesson 7: Performance and Display Issues**

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

## **Lesson 8: Drawing References and Comparison**

- Reusing a Drawing File
- Changing Drawing References
- Using Draw Compare
- SOLIDWORKS Design Checker

