

Course: SOLIDWORKS Practitioner

MFG

Description

Learn how to create parts, assemblies and drawings. This premium 3D course teaches you how to use SOLIDWORKS mechanical design and automation software to build and draw parametric models of parts and assemblies. This is a 5 day course. Day 1-3 is dedication to Parts and Assemblies. Day 4-5 Drawings.

Training objectives

On completion of this course you will be able to create SOLIDWORKS parts, build assemblies and create production drawings and advanced drawings of parts and assemblies.

Prerequisites

Mechanical design experience. Experience with Windows Operating System.

Completed the online tutorials that are integrated in the SOLIDWORKS software. You can access the online tutorials by clicking Help, Online Tutorial.

Skills you will acquire

You will be able to create SOLIDWORKS parts, build assemblies, create production drawings and customise drawing sheets.

Who should attend

New CAD users, seasoned users of other CAD systems and veteran SOLIDWORKS users can all benefit from this training.

Delivery mode



Face-to-face



Virtual Classroom

Duration



5 days



Course Outline

STAGE 1

Lesson 1: SOLIDWORKS Basics and the User Interface

- What is the SOLIDWORKS Software?
- Design Intent
- File Reference
- Opening Files
- The SOLIDWORKS User Interface
- Using the Command Manager

Lesson 2: Introduction to Sketching

- 2D Sketching
- 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
- Centre Marks
- Dimensioning
- Changing Parameters

Lesson 4: Drawing Sheets and Views

- Sheets and Formats Terminology
- Drawing Views

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

Lesson 5: Dimensions

- Manipulating
- Dimensions
- Dimension Properties

Lesson 6: 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 7: Patterning

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

Lesson 8: 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 9: Shelling and Ribs

- Shelling and Ribs
- Analyzing and Adding Draft
- Shelling
- Ribs
- Full Round Fillets
- Thin Features

Lesson 10: Editing/Repairs

- Part Editing
- Editing Topics
- Sketch Issues
- Freezing
- Features FilletXpert

Lesson 11: Editing/Design Changes

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

Lesson 12: Configurations

- Configurations
- Using Configurations
- Other Methods to Create Configurations Using Global Variables and Equations
- Creating Equalities
- Global Variables Defining the Overall Width Equations
- Creating a Minimum Edge Distance Modeling Strategies for Configurations
- Editing Parts that Have Configurations
- Design Library

Lesson 13: Annotations

- Adding Annotations
- Annotation Types
- Blocks

STAGE 2

Lesson 14: Sheet Formats and Templates

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

Lesson 15: Bottom-Up Assembly Modeling

- Case Study: Universal Joint Bottom-Up 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 16: Using Assemblies

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

Lesson 17: Assembly

- Drawing Views
- Creating Views of Assemblies

Lesson 18: 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
- Tables In the Drawing

Lesson 19: Performance and Display Issues

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

Lesson 20: Drawing References and Comparison

- Reusing a Drawing File
- Changing Drawing References
- Using DrawCompare SOLIDWORKS Design Checker