

SOLIDWORKS Essentials



Overview:

The course will teach you how to use the SOLIDWORKS mechanical design automation software to build parametric models of parts and assemblies, and creating drawings of parts and assemblies. Some basic and more advanced features will be covered.



Duration:

InClass: 4 Days (Full Time) 8:30am- 4:30pm

Distance Learning: 7 Days (Part Time)



Pre-requisites:

- Complete / attend Introduction to Technical Drawing course or similar experience.
- Computer literacy skills.
- Mechanical design experience

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
- Basic sketching
- Rules that govern sketches
- Design intent
- Sketch relations
- Dimensions
- Extrude
- Sketching guidelines

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
- View Selector
- Using the hole wizard
- Filleting
- Editing tools
- Detailing basics
- Drawing views
- Centre Marks
- Dimensioning
- Changing parameters

Symmetry and Draft

- Case study
- 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 patterns
- Circular patterns
- Reference geometry
- Planes
- Mirror patterns
- Using pattern seed only
- Up to reference
- Sketch driven patterns

Revolved Features

- Design intent
- Revolved features
- Building the rim
- Building the spoke
- Edit material



- Mass properties
- Files properties
- SOLIDWORKS Simulation Xpress
- Using Simulation Xpress
- The Simulation Xpress interface

Shelling and Ribs

- Shelling and Ribs
- Analysing and adding draft
- Other draft options
- Shelling
- Ribs
- Full round fillets
- Thin features

Editing: Repairs

- Part editing
- Editing topics
- Sketch issues

Editing: Design Changes

- Part editing
- Design changes
- Information from a model
- Rebuilding tools
- Sketch contours
- Replace sketch entity

Configurations

- Configurations
- Using configurations
- Other methods to create configurations
- Modelling strategies for configurations
- Editing parts that have configurations
- Design library

Global Variables and Equations

- Using global variables and equations
- Renaming features and dimensions
- Design rules using global variables
- Global variables
- Equations
- Using operators and functions

Using Drawings

- Removed sections
- Detail views
- Drawing sheets and sheet formats
- Model views



- Section view
- Annotations

Bottom-Up Assembly Modelling

- 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
- Subassemblies
- Smart mates
- Inserting subassemblies
- Pack and Go feature

Using Assemblies

- Using assemblies
- Analysing the assembly
- Checking for clearances
- Changing the values of dimensions
- Exploded assemblies
- Rollback and reorder explode steps
- Explode line sketch
- Bill of materials
- Assembly drawings

