Course details:

This course provides you with your introduction to SOLIDWORKS mechanical design software. The course starts with the elementary tools needed to create basic part models and completes with the assembling and evaluating a fully working mechanism along with creating the drawings needed to detail your designs. No matter what your background this course will provide you with the fundamentals needed to go away and start working in the software. Principally you will learn how to create robust designs, which can stand up to numerous design changes that are required in a design cycle.

1: Basics and the User Interface

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

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
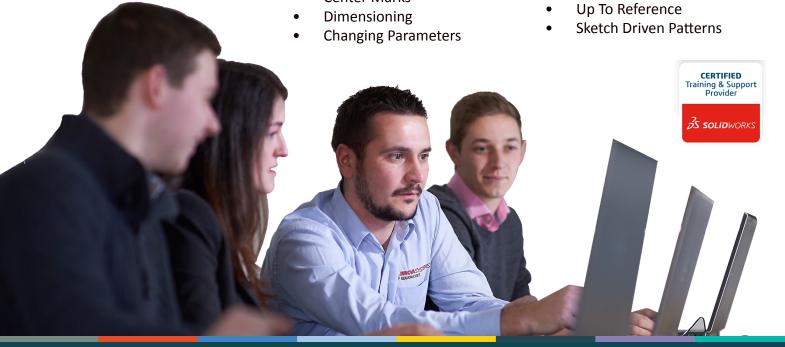
3: Basic Part Modeling

- 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
- Filleting
- Editing Tools
- Detailing Basics
- Drawing Views
- Center Marks
- Dimensioning
- Changing Parameters

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
- Copy and Paste Features

5: Patterning

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

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6: Revolved Features

- Case Study: Handwheel
- Design Intent
- Revolved Features
- Building the Rim
- Building the Spoke
- Edit Material
- Mass Properties
- File Properties
- SimulationXpress

7: Shelling and Ribs

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

8: Editing: Repairs

- Part Editing
- Editing Topics
- Sketch Issues

9: Editing: Design Changes

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

10: Configurations

- Configurations
- Using Configurations
- Other Methods to Create Configurations

- Modeling Strategies for Configurations
- Editing Parts that Have Configurations
- Design Library
- In the Advanced Course

11: Global Variables and Equations

- Using Global Variables and Equations
- Renaming Features and Dimensions
- Design Rules Using Global Variables and Equations
- Global Variables
- Equations
- Using Operators and Functions

12: Using Drawings

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

13: 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
- Subassemblies
- Smart Mates
- Inserting Subassemblies
- Pack and Go

14: Using Assemblies

- Using Assemblies
- Analyzing 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