



# SOLIDWORKS ESSENTIALS

## Who should attend:

Application Engineer  
R&D Engineer  
Product Designer or Engineer  
Industrial Engineer

## Prerequisites:

Experience with the Windows™ Operating System

Basic understanding of part design and drafting

Mechanical Design Experience

## Duration:

5 Days

## Methodology:

Practical hands-on with using computers, lecturing, discussions and case studies

## Introduction

By quickly getting up to speed, as well as using best modelling practices, the SOLIDWORKS Essentials course ultimately ensures maximum efficiency and productivity for all SOLIDWORKS users. New CAD users, seasoned users of other CAD systems and veteran SOLIDWORKS users can all benefit from Essentials training. Real life case studies are used to illustrate processes, teaching everything you need to know for building parametric models of parts and assemblies, and making drawings of those parts and assemblies

## Objective

At the end of this program participants are expected to:

- Create parametric sketches
- Create parts and build assemblies
- Create and control different versions of your parts with configurations
- Automate design modification with equations and excel design tables
- Create drawings of your designs
- [EXTRA] Digital technical drawing presentation

## IME CADCAM TRAINING CENTRE SDN BHD

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Key Topics

**Day 1**

**Course Outline**

**Lesson 1 : SOLIDWORKS BASICS and the User Interface**

- What is the SOLIDWORKS Software?
- File References
- Opening Files
- 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 Modeling
- Terminology
- Choosing the Best Profile
- Choosing the Sketch Plane
- Details of the Part
- Boss Feature
- Sketching on a Planar Face
- Cut Feature
- View Selectors
- Using the Hole Wizard
- Filleting
- Editing Tools
- 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
- Copy and Paste Features

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Key Topics

**Day 2**

**Lesson 5 : Patterning**

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

**Lesson 6 : 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 7 : Shelling and Ribs**

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

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Day 3

**Lesson 8 : Editing : Repairs**

- Part Editing
- Editing Topics
- Sketch Issues

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

**Lesson 11 : Global Variables and Equations**

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

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Day 4

**Lesson 12 : Using Drawings**

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

**Lesson 13 : Bottom-Up Assembly Modelling**

- 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

**Lesson 14 : Using Assemblies**

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

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Day 5

**Lesson 15 : Assembly Drawing Views**

- Assembly Drawing Views
- Creating Views of Assembly

**Lesson 16 : Sheet Format 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 17 : Performance and Display Issues**

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

**Lesson 18 : DimXpert and Paperless Drawing**

- Understand DimXpert Settings in Document Properties.
- Create DimXpert annotations using the Auto Dimension Scheme tool
- Modify DimXpert annotations
- Understand the DimXpertManager
- Manually create DimXpert features and annotations.

**Lesson 19 : Mobile presentation**

- Experiencing the Mobile Presentation

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