

SOLIDWORKS Design Essentials Course Outline

 DURATION/TIME 5 Days 9:00 am - 5:00 pm	 METHODOLOGY Practical hands-on with using computers, lecturing, discussions and case studies.
 PREREQUISITES Mechanical design experience; experience with the Windows® operating system	 TARGET Application Engineer, R&D Engineer, Product Designer or Engineer, and Industrial Engineer

INTRODUCTION

SOLIDWORKS Design Essentials teaches you how to use the SOLIDWORKS Design mechanical automation software to build parametric models of parts and assemblies, and how to make 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.

CONTACT

IME CADCAM RESOURCE CENTRE SDN BHD
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SOLIDWORKS Design Essentials

DAY 1

Course Outline:

Lesson 1: SOLIDWORKS Design Basics and User Interface

- What is the SOLIDWORKS Design Software?
- Design Intent
- File References
- Opening Files
- The SOLIDWORKS Design 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
- Sharing a Model
- 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
- 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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DAY 2

Course Outline:

Lesson 5: Patterning

- Why Use Patterns?
- Linear Patterns
- Circular Patterns
- Reference Geometry
- Planes
- Mirror Patterns
- Coordinate System
- Symmetric Patterns
- 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

Course Outline:

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
- Sketch Contours
- Creating a Design Revision
- Replace Sketch Entity

Lesson 10: Configurations

- Configurations
- Using Configurations
- Other Methods to Create Configurations
- Modeling Strategies for Configurations
- Editing Parts that Have Configuration
- Design Library

Lesson 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

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Day 4

Course Outline:

Lesson 12: Using Drawings

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

Lesson 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

Lesson 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

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Day 5

Course Outline:

SOLIDWORKS DRAWINGS – ISO

Lesson 1: Introduction to Drawing Templates

- Structure of a Drawing Document
- Drawing Document, Drawing Sheet, and Drawing Templates
- Case Study: Setting a Default Location for Templates
- Case Study: Understanding Drawing Templates
- Case Study: Designing Your Own Drawing Template

Lesson 2: Sheet Format Customization

- Case Study: Custom Sheet Format

Lesson 3: Using the Custom Sheet Format File

- Case Study: Saving the Sheet Format File

Lesson 4: Advanced Drawing Views

- Case Study: Auxiliary View and Crop View
- Case Study: Assembly Drawing Views
- Case Study: Custom View Orientation

Lesson 5: Drawing Performance

- Case Study: Improving Performance of a Drawing

Test and Assessments

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