3D Solid Modeling Design

Duration : 5 days
Time : 9:00am – 5:00pm
Methodology : Practical hands-on with using computers, lecturing, discussions and case studies
Target : Application Engineer
R&D Engineer
Product Designer or Engineer
Industrial Engineer

Prerequisite : To ensure a consistent learning experience for all students, participants should have:
• Mechanical Design Experienced
• Experience with the Windows™ Operating System.
• Basic Understanding of part design and drafting

Objective : The SolidWorks Essentials course will discuss the fundamental solid modeling technique involve in product design. This will include:
• Understanding and applying the product design intent for creating a solid model.
• Modeling the product desired outlook by using different modeling approach.
• Designing casting products such as a ratchet body by considering the manufacturing aspect.
• Learn how to validate and improve an existing product design.
• Strengthening the consumer product design by adding ribs such as a multiple extension plug.
• Interpreting and resolving common modeling errors during detail design stage.
• Learn to redesign an existing product design efficiently.
• Exposed to the different techniques for creating a series of product.
• Building the final project model to demonstrate the desired mechanism.
• Learn about basic drafting technique to represent different view of a product design.
• Applying different types of corner to improve the appearance of your product design.
• Creating different type of drawing views with dimensions and annotations for documenting the solid model design intent.
• Prepare standard drawing templates according to company standard for reuse in different projects.
• Prepare complete Bill of Material templates with custom properties for easy documentation for a project models.
• Introduce new window interfaces for comparing revised drawings and checking drawing standards.

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Course Outline:

Day 1
Lesson 1: SolidWorks Basics and The User Interface
- What is the SolidWorks Software?
- Design Intent
- File References
- Opening Files
- The SolidWorks User Interface

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 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
- Using the Hole Wizard
- View Options
- Filleting
- Detailing Basics
- Drawing Views
- Center Marks
- Dimensioning
- Changing Parameters

Lesson 4: Modeling a Casting or Forging
- 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
- Using Copy and Paste

Day 2
Lesson 5: Patterning
- Why Use Patterns?
- Reference Geometry
- Linear Pattern
- Circular Pattern
- Mirror Pattern
- Using Pattern Seed Only
- Sketch Driven Pattern

Lesson 6:
- Case Study: Handwheel
- Design Intent
- Revolved Features
- Building The Rim
- Building The Spoke
- Edit Material
- Mass Properties
- File Properties
- SolidWorks Simulation Xpress
- Using SolidWorks Simulation Xpress
- The Simulation Xpress Interface
Lesson 7: Shelling and Ribs
- Shelling and Ribs
- Analyzing and Adding Draft
- Other Options for Draft
- Shelling
- Ribs
- Full Round Fillets
- Thin Features

Day 3
Lesson 8: Editing: Repairs
- Part Editing
- Editing Topics
- Sketch Issues
- FilletXpert
- DraftXpert

Lesson 9: Editing: Design Changes
- Part Editing
- Design Changes
- Information From a Model
- Rebuilding Tools
- Sketch Contours
- Editing with Instant 3D

Lesson 10: Configurations
- Configurations
- Using Configurations
- Creating Configurations
- Link Values
- Equations
- Configure Dimension/Feature
- Modeling Strategies for Configurations
- Editing Parts that Have Configurations
- Design Library

Day 4
Lesson 11: Using Drawings
- More About Making Drawings
- Section View
- Model Views
- Broken Views
- Detail Views
- Drawing Sheets and Sheets Formats
- Projected Views
- Annotations

Lesson 12: Bottom-Up Assembly Modeling
- Case Study: Universal Joint
- Bottom-Up Assembly
- Creating a New Assembly
- Position of the First Component
- Feature Manager Design Tree and Symbols
- Adding Components
- Using Part Configurations in Assemblies
- Sub-assemblies
- Smart Mates
- Inserting Sub-assemblies
- Pack and Go

Lesson 13: Using Assembly
- Using Assemblies
- Analyzing the Assembly
- Checking for Clearances
- Changing the Values of Dimensions
- Exploded Assemblies
- Exploded Line Sketch
- Bill of Materials
- Assembly Drawings
Day 5
Lesson 14: Assembly Drawing Views
- Assembly Drawing Views
- Creating Views of Assembly

Lesson 15: Sheet Formats and Templates
- Sheet Formats and Templates
- Drawing Templates
- Properties in the Template
- User Defined Properties
- Customizing a Sheet Format
- Import Legacy Data
- Define Title Block
- Updating Sheet Formats

Lesson 16: Performance and Display Issues
- Performance and Display Issues
- Large Assembly Mode
- Lightweight Drawings
- Detached Drawings
- Display Issues in Drawing Views

Lesson 17: Drawing References and Comparison
- Reusing a Drawing File
- Changing Drawing References
- Using DrawCompare
- SolidWorks Design Checker
- Updating Sheet Formats