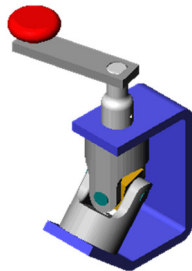
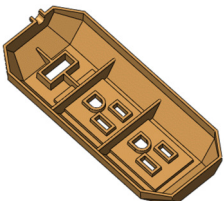
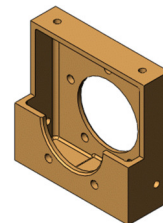
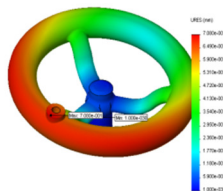


## 3D Solid Modeling Design

- 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



Model Name: 001\_000001\_01  
Date: 01/01/2010 10:00:00 AM  
Author: 001\_000001\_01  
Description: 001\_000001\_01



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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
- Filletting
- 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

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## 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

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**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

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