SOLIDWORKS Simulation Associate (CSWA-S) Certification Prep Course

Target Audience

This course is designed for engineering students, CAD professionals, and early career engineers preparing for the **SOLIDWORKS Simulation Associate Certification Exam**. It is ideal for learners who already have a foundation in SOLIDWORKS modelling, or even who are starting fresh in FEA and want to demonstrate competency in simulation fundamentals, including FEA workflows, meshing, boundary conditions, and result interpretation.

Course Outcomes

By the end of this course, participants will be able to:

- Understand the fundamentals of SOLIDWORKS Simulation and Finite Element Analysis (FEA).
- Build mathematical and finite element models for parts and assemblies.
- Apply fixtures, loads, and material properties correctly.
- Generate and refine meshes for accuracy.
- Interpret simulation outputs including stresses, displacements, and safety factors.
- Perform assembly analysis with interactions and connectors.
- Apply advanced meshing techniques (shells, beams, mixed meshes).
- Conduct design studies and thermal stress analysis.
- Validate results against analytical methods and generate reports.
- Be fully prepared for the **SOLIDWORKS Simulation Associate Certification Exam**.

Course Objectives

- Provide a structured introduction to SOLIDWORKS Simulation aligned with certification requirements.
- Train learners in preprocessing, meshing, solving, and postprocessing workflows.
- Develop proficiency in handling assemblies, connectors, and boundary conditions.
- Enable learners to analyze thin components, beams, and mixed meshes.
- Teach advanced topics including symmetry, bonded mesh options, and thermal stress analysis.
- Reinforce learning through case studies and exercises similar to certification exam problems.

Course Outline

The course comprises **40 hours** of theory and labs and is divided into **14 different chapters**. Each chapter will be followed by hands-on lab exercises to reinforce learning and gauge understanding of the topics covered.

Table of Contents:

Introduction / Getting Started

- Introduction to SOLIDWORKS Simulation
- Course Overview
- Simulation Workflow
- Types of Analysis
- Finite Element Method (FEM) Basics
- Terminology

Lesson 1: Basic Static Analysis

- Objectives
- Introduction to Static Analysis
- Steps in Analysis Procedure

Case Study

- Project Description
- Stages in the Process

Pre-processing

- Material Assignment
- Fixtures (Boundary Conditions)
- External Loads

Meshing

- Mesh Generation
- Mesh Types
- Mesh Density

Solving

- Running the Simulation

Post-processing

- Displacement Results
- Stress Results (Von Mises)
- Factor of Safety

Validation

- Checking Results
- Mesh Convergence Basics

Wrap-up

- Summary
- Questions

Exercises

- Exercise 1: Basic Part Analysis

- Exercise 2: Stress Analysis
- Exercise 3: Container Handle

Lesson 2: Mesh Controls, Stress Concentrations and Boundary Conditions

- Objectives
- Mesh Control
- Case Study: The L Bracket
  - Project Description
  - Stages in the Process
  - Run This Study
- Analysis with Local Mesh Refinement
- Mesh Controls
- Results
- Results Comparison
- Stress Singularities
- Suppressed Configuration
- Case Study: Analysis of Bracket with a Fillet
- Result Force
- Case Study: Analysis of a Welded Bracket
- Understanding the Effect of Boundary Conditions
- Conclusion
- Summary
- Questions

Exercises

- Exercise 4: C-bracket
- Exercise 5: Bone Wrench

Lesson 3: Assembly Analysis with Interactions

- Objectives
- Interaction Analysis
- Case Study: Pliers with Global Interaction
  - Project Description
  - Stages in the Process
- Component Interaction
- Component Interaction: Options
- Component Interactions: Default Setting
- Component Contact: Hierarchy and Conflicts
- Study Properties
- Viewing Assembly Results
- Strong Enough?
- Contact or Bonded Interaction
- Pliers with Local Interaction
  - Local Interaction

    - Local Interaction Types
    - Self-Contact
    - Required Force
    - Local Interaction Contact Properties
    - Contact Formulation
    - Contact Stresses
- Summary
- Questions

Exercise

- Exercise 6: Two Ring Assembly

Lesson 4: Symmetrical and Free Self-Equilibrated Assemblies

- Objectives
- Shrink Fit Parts
- Case Study: Shrink Fit
    - Project Description
    - Symmetry
    - Stages in the Process
- Defeaturing
- Rigid Body Mode
- Underconstrained Bodies
- Automatically Detect Underconstrained Bodies
- Shrink Fit Contact Condition
- Plot Results in Local Coordinate System
- Cylindrical Coordinate Systems
- Saving All Plots
- What's Wrong Feature
- Analysis with Soft Springs
    - Soft Springs
    - Inertial Relief
    - Collapse Tree Item
- Summary

Exercise

- Exercise 7: Bone Wrench Assembly

Lesson 5: Assembly Analysis with Connectors and Mesh Refinement

- Objectives
- Case Study: Cardan Joint
    - Problem Statement
- Remote Load/Mass
- Load Location
- Reference Coordinate System
- Types of Loads
- Connection Type

- Connectors
  - Connector Types
  - Automatic Conversion of Toolbox Fasteners to Bolts
  - Distributed Coupling
  - Bolt Strength Data
  - Bolt Pre-load
  - Bolt Tight fit and Diameter
- Interaction Viewer
- Automatically Find Local Interactions
- Local Interaction Contact Properties
- Pin Connectors
- Rotational and Axial Stiffness
- Virtual Wall, Axial and Tangential Stiffness
- Mesh Control in an Assembly
- Required Number of Solid Elements in Thin Features
- Mesh Plots
  - Quality Plot
  - Pin/Bolt Force
  - Factor of Safety Plot
- Summary

Exercises

- Exercise 8: Chain Link
- Exercise 9: Lift Assembly
- Bearings
  - Bearing Connectors
  - Bearing Fixtures
  - Bearing Loads
- Exercise 10: Spot Welds – Solid Mesh
- Interaction Between Parts
- Exercise 11: Bolt Connectors
- Exercise 12: Wood Splitter

Lesson 6: Bonded Mesh Options

- Objectives
- Bonded Mesh Options
- Case Study: Rotor
  - Project Description
- Centrifugal Force
- Cyclical Symmetry
- Bonding Options
  - Gap Range
  - Common Nodes
  - Automatic Switch
- Bonding Formulation
  - Node to Surface

- Surface to Surface
- Summary

Exercise

- Exercise 13: Vise Grip Pliers

Additional

- Spring Connector Types
- Spring Connector Options

Lesson 7: Analysis of Thin Components

- Objectives
- Thin Components
- Case Study: Pulley
  - Project Description
- Part 1: Mesh with Solid Elements
  - Symmetry Fixtures
- Part 2: Refined Solid Mesh
- Solid vs Shell
- Creating Shell Elements
- Part 3: Shell Elements – Mid-plane Surface
- Thin vs Thick Shells
- Shell Mesh Colors
- Changing Mesh Orientation
- Shell Element Alignment
- Render Shell Thickness in 3D
- Applying Symmetry Restraints
- Results Comparison
- Computational Effort
- Case Study: Joist Hanger
  - Project Description
- Convergence Check Plot
- Summary
- Questions

Exercises

- Exercise 14: Bracket
- Exercise 15: Shell Mesh Using Outer/Inner Faces
- Exercise 16: Edge Weld Connector
- Exercise 17: Container Handle Weld

Lesson 8: Mixed Meshing Shells & Solids

- Objectives
- Mixed Meshing Solids and Shells
- Bonding Shells and Solids
- Mixed Mesh: Supported Analysis Types
- Case Study: Pressure Vessel
  - Project Description
- Analyze the Assembly

- Preparing the Model
- Material
- Steel Identification Systems
    - UNS Index
    - Other Indices
- Bulk and Shear Moduli
- Shell to Shell Bonding
- Shell to Solid Bonded
- Failure Diagnostics
- Meshing Small Features
- Incremental Meshing
- Summary
- Questions

Exercise

- Exercise 18: Mixed Mesh Analysis

Additional

- Simulation Evaluator

Lesson 9: Beam Elements – Analysis of a Conveyor Frame

- Objectives
- Project Description
- Element Choices
    - Beam Elements
    - Truss Elements
- Stages in the Process
- Slenderness Ratio
- Section Properties
- Connected and Disconnected Joints
- Sphere Diameter Defining Beam Joint
- Beam Joints: Locations
- Beam Joint Types
- Render Beam Profile
- Beam Stress Components
- Cross-section Directions
- Bending Moment and Shear Force Diagrams
- Summary
- Questions

Lesson 10: Mixed Meshing Solids, Beams & Shells

- Objectives
- Mixed Meshing
- Case Study: Particle Separator
    - Project Description
    - Stages in the Process
- Beam Mesh

- Beam Imprint
- Summary **Exercises**
- Exercise 19: Cabinet
- Exercise 20: Frame Rigidity

Lesson 11: Design Study

- Objectives
- Design Study
- Case Study: Suspension Design
  - Project Description
  - Stages in the Process
- Part 1: Multiple Load Cases
- Design Studies
- Parameters
- Design Study Results
- Design Study Options
- Part 2: Geometry Modification
- Design Study Graph
- Summary

Exercise

- Exercise 21: Design Study

Lesson 12: Thermal Stress Analysis

- Objectives
- Thermal Stress Analysis
- Case Study: Bimetallic Strip
  - Project Description
- Material Properties
- Importing Temperature and Pressure
- Saving Model in its Deformed Shape
- Summary

Lesson 13: Adaptive Meshing

- Objectives
- Adaptive Meshing
- Case Study: Support Bracket
- Geometry Preparation
- h-Adaptivity Study
- h-Adaptivity Options
- h-Adaptive Plots
- Convergence Graph
- Review h-adaptive Solution
- Strain Energy Error is NOT Stress Error
- p-Adaptivity Study
- p-Adaptive Solution Method

- h vs p Elements
- Method Comparison
- h vs p Elements – Summary
- Which Solution Method is Better?
- Summary

Exercise

- Exercise 22: Mohr-Coulomb Failure

Additional

- Unique Load Profiles
- Relevant Results
- Adaptive Meshing Constraints

Lesson 14: Large Displacement Analysis

- Objectives
- Small vs Large Displacement Analysis
- Case Study: Clamp
- Part 1: Small Displacement Linear Analysis
- Results Discussion
- Contact in Small and Large Displacement Analyses
- Part 2: Large Displacement Nonlinear Analysis
- Permanent Deformation
- SOLIDWORKS Simulation Premium
- Summary
- Questions