SolidWorks Simulation Training Course

Course Duration: 3 days

This course is designed to make SolidWorks users productive more quickly with the SolidWorks Simulation Bundle. This course will provide an in-depth coverage on the basics of Finite Element Analysis (FEA), covering the entire analysis process from meshing to evaluation of results for parts and assemblies. The class discusses linear stress analysis, gap/contact analysis, and best practices.

Prerequisites : Knowledge of SolidWorks and basic mechanical engineering concepts is recommended.

Who should attend: All SolidWorks Simulation users wishing to create better designs in SolidWorks by performing analysis and evaluating the behaviour of their parts and assemblies under actual service conditions.

Introduction to FEA
About This Course
What is SolidWorks Simulation?
What Is Finite Element Analysis
Build Mathematical Mode
Build Finite Element Mode
Solve Finite Element Mode
Analyze Result
Errors in FEA
Finite Elements
Degrees of Freedom
Calculations in FEA
Interpretation of FEA Results
Units of Measurement
Limitations of SolidWorks Simulation

Lesson 1: The Analysis Process
Objectives
The Analysis Process
Case Study: Stress in a Plate
Project Description
SolidWorks Simulation Options
Preprocessing
Meshing
Postprocessing
Multiple Studies
Reports

Lesson 2: Mesh Controls, Stress Concentrations and Boundary Conditions
Objectives
Mesh Control
Case Study: The L Bracket
Project Description
Case Study: Analysis of Bracket with a Fillet
Case Study: Analysis of a Welded Bracket
Understanding the Effect of Boundary Conditions

Lesson 3: Assembly Analysis with Contacts
Objectives
Contact Analysis
Case Study: Pliers with Global Contact
Pliers with Local Contact

Lesson 4: Symmetrical and Free Self-Equilibrated Assemblies
Objectives
Shrink Fit Parts
Case Study: Shrink Fit
Project Description
Analysis with Soft Springs

Lesson 5: Assembly Analysis with Connectors
Objectives
Connecting Components
Connectors
Case Study: Vise Grip Pliers

Lesson 6: Compatible/Incompatible Meshes
Objectives
Compatible / Incompatible Meshing
Case Study: Rotor

Lesson 7: Assembly Analysis Mesh Refinement
Objectives
Mesh Control in an Assembly
Case Study: Cardan Joint
Problem Statement
Part 1: Draft Quality Coarse Mesh Analysis
Part 2: High Quality Mesh Analysis

Lesson 8: Analysis of Thin Components
Objectives
Thin Components
Case Study: Pulley
Part 1: Mesh with Solid Elements
Part 2: Refined Solid Mesh
Solid vs. Shell
Creating Shell Elements
Part 3: Shell Elements - Mid-plane Surface
Results Comparison
Case Study: Joist Hanger

Lesson 9: Mixed Meshing Shells & Solids
Objectives
Mixed Meshing Solids and Shells
Case Study: Pressure Vessel

Lesson 10: Mixed Meshing Solids, Beams & Shells
Objectives
Mixed Meshing
Case Study: Particle Separator

Lesson 11: Design Scenarios
Objectives
Design Study
Case Study: Suspension Design
Part 1: Multiple Load Cases
Part 2: Geometry Modification

Lesson 12: Thermal Stress Analysis
Objectives
Thermal Stress Analysis
Case Study: Bimetallic Strip
Examining Results in Local Coordinate Systems
Saving Model in its Deformed Shape

Lesson 13: Adaptive Meshing
Objectives
Adaptive Meshing
Case Study: Support Bracket
h-Adaptivity Study
p-Adaptivity Study
h vs. p Elements - Summary

Lesson 14: Large Displacement Analysis
Objectives
Small vs. Large Displacement Analysis
Case Study: Clamp
Part 1: Small Displacement Linear Analysis
Part 2: Large Displacement Nonlinear Analysis

SOLIDWORKS Certified Training & Support Provider
SOLIDWORKS Certified Documentation Services Provider
SOLIDWORKS Authorized Reseller

Contact Details

North Wales Office
CCSL | Unit 8 Old Marsh Farm Barns | Welsh Road | Sealand | Deeside | CH5 2LY
Phone: 01244 289350

South Wales Office
CCSL | Enterprise House | Navigation Park | Abercynon | CF45 4SN
Phone: 01443 719191

www.ccsl-cad.co.uk
Training@ccsl-cad.co.uk