Digital Product Simulation

Training SIMULIA
Analysis of Composites Materials with ABAQUS

General information

- ➢ Duration — 2 days
- ➢ Language — French or English
- ➢ Participant profiles — Structural Engineer
- ➢ Prerequisite — ABAQUS Standard Initiation

Overall objectives

Composite materials are used in many design applications because of their high stiffness-to-weight ratios. This seminar shows you how to use Abaqus effectively to model composite materials

DAY 1	
Topics	*Description*
Introduction	• Anisotropic Elasticity
Macroscopic Modeling	• Viscoelasticity
Mixed Modeling	• Thermal Expansion • Laminated Composite Shells • Continuum Shell Elements • Continuum Shell Meshing • Continuum Solid Elements • Symmetry Conditions and Laminated Structures
Composites modeling with Abaqus	• Understanding Composite Layups • Understanding Composite Layup Orientations • Defining Composite Layup Output • Viewing a Composite Layup • Abaqus/CAE Demonstration: Three-ply composite • Composites Modeler for Abaqus/CAE • Workshops

DAY 2	
Topics	*Description*
Reinforcement Modeling	• Rebar Layers
Modeling of Sandwich Composites	• Embedded Elements
	• Introduction to Sandwich Composites • Abaqus Usage • Modeling Skins with Abaqus/CAE • Stiffened Composite Panels • Abaqus Usage
	• Failure Criteria in Laminates • Failure Theories • Progressive Damage of Fiber-Reinforced Composites
Modeling of Stiffened Panels	
Damage and Failure in Composites	