Learning Module

Analysis of Composite Materials with Abaqus

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.

Objectives

Upon Completion Of This Course You Will Be Able To:
- Define anisotropic elasticity for combining the fibermatrix response.
- Define composite layups.
- Model progressive damage and failure in composites.
- Model delamination and low-cycle fatigue of composite structures.
- Model sandwich composite structures and stiffened composite panels.

Knowledge Prerequisites

This course is recommended for engineers with experience using Abaqus.

Language(s) for selected release

English

Brands

Simulia

Available Releases

SIMULIA 2021, SIMULIA 2020, SIMULIA 2019, SIMULIA 2018, SIMULIA 2017, SIMULIA 2016, SIMULIA V6.14, SIMULIA V6.13, SIMULIA V6.12

Duration

24 hours

Discipline

Advanced Abaqus

Contents

Overview - Analysis of Composite Materials with Abaqus
1 - Introduction
2 - Macroscopic Modeling
3 - Laminate Modeling
4 - Composite Modeling with Abaqus
5 - Modeling Damage and Failure in Composites
6 - Cohesive Behavior
7 - Virtual Crack Closure Technique (VCCT)
8 - Reinforcement Modeling
9 - Modeling of Sandwich Composites
10 - Modeling of Stiffened Panels
11 - Fatigue Crack Growth at Material Interfaces
A1 - Debond Capability
A2 - Cohesive Element Modeling Techniques
A3 - More on Continuum Shell Elements
A4 - Alternative Modeling Techniques
A5 - Modeling Composite Material Impact
A6 - Material Orientation Examples
A7 - Multiscale Modeling