Analysis of Composite Materials with Abaqus

Abaqus 2020
Course objectives
Upon completion of this course you will be able to:

- Define anisotropic elasticity for combining the fiber-matrix response
- Define composite layups
- Model progressive damage and failure in composites
- Model delamination and fatigue crack growth of composite structures
- Model sandwich composite structures and stiffened composite panels

Targeted audience
Simulation Analysts

Prerequisites
This course is recommended for engineers with experience using Abaqus
Day 1

- **Lesson 1**  Introduction
- **Lesson 2**  Macroscopic Modeling
- **Lesson 3**  Laminate Modeling
  - **Workshop 1**  The Pagano Plate Problem
- **Lesson 4**  Composite Modeling with Abaqus
  - **Workshop 2a**  Buckling of a Laminate Panel
  - **Workshop 2b**  Composite Wing Section
  - **Workshop 3**  Composite Yacht Hull (Optional)
Day 2

- Lesson 5  Modeling Damage and Failure in Composites
- Lesson 6  Cohesive Behavior
  - Workshop 4  Analysis of a DCB using Cohesive Behavior
- Lesson 7  Virtual Crack Closure Technique (VCCT)
  - Workshop 5  Analysis of a DCB using VCCT (Abaqus/Standard)
  - Workshop 6  Analysis of a DCB using VCCT (Abaqus/Explicit)
Day 3

- Lesson 8  Reinforcement Modeling
- Lesson 9  Modeling of Sandwich Composites
  - Workshop 7  Bending of a Sandwich Beam
- Lesson 10  Modeling of Stiffened Panels
  - Workshop 8  Bending of a Reinforced Flat Panel under Uniform Pressure
- Lesson 11  Fatigue Crack Growth at Material Interfaces
  - Workshop 9  Fatigue Crack Growth in a DCB Specimen
Additional Material

- Appendix 1: Debond Capability
- Appendix 2: Cohesive Element Modeling Techniques
- Appendix 3: More on Continuum Shell Elements
- Appendix 4: Alternative Modeling Techniques
- Appendix 5: Modeling Composite Material Impact
- Workshop 10: Perforation of a Composite Plate
- Appendix 6: Material Orientation Examples
- Appendix 7: Multiscale Modeling
SIMULIA is the Dassault Systèmes brand for Realistic Simulation solutions

- Portfolio of established, best-in-class products
  - Abaqus, Isight, Tosca, fe-safe, Simpack

* Included in extended licensing pool
## SIMULIA’s Power of the Portfolio

### Abaqus
- Routine and Advanced Simulation
- Linear and Nonlinear, Static and Dynamic
- Thermal, Electrical, Acoustics
- Extended Physics through Co-simulation
- Model Preparation and Visualization

### Isight
- Process Integration
- Design Optimization
- Parametric Optimization
- Six Sigma and Design of Experiments

### Tosca
- Non-Parametric Optimization
- Structural and Fluid Flow Optimization
- Topology, Sizing, Shape, Bead Optimization

### fe-safe
- Durability Simulation
- Low Cycle and High Cycle Fatigue
- Weld, High Temperature, Non-metallics

### Simpack
- 3D Multibody Dynamics Simulation
- Mechanical or Mechatronic Systems
- Detailed Transient Simulation (Offline and Realtime)

## Realistic Human Simulation
- High Speed Crash & Impact
- Noise & Vibration

## Material Calibration
- Workflow Automation
- Design Exploration

## Conceptual/Detailed Design
- Weight, Stiffness, Stress
- Pressure Loss Reduction

## Safety Factors
- Creep-Fatigue Interaction
- Weld Fatigue

## Complete System Analyses
- (Quasi-)Static, Dynamics, NVH
- Flex Bodies, Advanced Contact
Join the Community!

How can you maximize the robust technology of the SIMULIA Portfolio?
Connect with peers to share knowledge and get technical insights

Go to www.3ds.com/slc to log in or join!

Let the SIMULIA Learning Community be Your Portal to 21st Century Innovation
Discover new ways to explore how to leverage realistic simulation to drive product innovation. Join the thousands of Abaqus and Isight users who are already gaining valuable knowledge from the SIMULIA Learning Community.

For more information and registration, visit 3ds.com/simulia-learning. Connect. Share. Spark Innovation.
SIMULIA Training

http://www.3ds.com/products-services/simulia/services/training-courses/

SIMULIA SERVICES
PROVIDING HIGH QUALITY SIMULATION AND TRAINING SERVICES TO ENABLE OUR CUSTOMERS TO BE MORE PRODUCTIVE AND COMPETITIVE.

Training Schedule & Registration
We offer regularly scheduled public seminars as well as training courses at customer sites. An extensive range of courses are available, ranging from basic introductions to advanced courses that cover specific analysis topics and applications. On-site courses can be customized to focus on topics of particular interest to the customer, based on the customer's prior specification. To view the worldwide course schedule and to register for a course, visit the links below.

North American
› By Location
› By Course

International
› By Location
› By Course

Live Online Training
› Full Schedule
The software described in this documentation is available only under license from Dassault Systèmes or its subsidiaries and may be used or reproduced only in accordance with the terms of such license.

This documentation and the software described in this documentation are subject to change without prior notice.

Dassault Systèmes and its subsidiaries shall not be responsible for the consequences of any errors or omissions that may appear in this documentation.

No part of this documentation may be reproduced or distributed in any form without prior written permission of Dassault Systèmes or its subsidiaries.

© Dassault Systèmes, 2019

Printed in the United States of America.

Abaqus, the 3DS logo, and SIMULIA are trademarks or registered trademarks of Dassault Systèmes or its subsidiaries in the US and/or other countries.

Other company, product, and service names may be trademarks or service marks of their respective owners. For additional information concerning trademarks, copyrights, and licenses, see the Legal Notices in the SIMULIA User Assistance.
<table>
<thead>
<tr>
<th>Lesson 1</th>
<th>11/19</th>
<th>Updated for Abaqus 2020</th>
</tr>
</thead>
<tbody>
<tr>
<td>Lesson 2</td>
<td>11/19</td>
<td>Updated for Abaqus 2020</td>
</tr>
<tr>
<td>Lesson 3</td>
<td>11/19</td>
<td>Updated for Abaqus 2020</td>
</tr>
<tr>
<td>Lesson 4</td>
<td>11/19</td>
<td>Updated for Abaqus 2020</td>
</tr>
<tr>
<td>Lesson 5</td>
<td>11/19</td>
<td>Updated for Abaqus 2020</td>
</tr>
<tr>
<td>Lesson 6</td>
<td>11/19</td>
<td>Updated for Abaqus 2020</td>
</tr>
<tr>
<td>Lesson 7</td>
<td>11/19</td>
<td>Updated for Abaqus 2020</td>
</tr>
<tr>
<td>Lesson 8</td>
<td>11/19</td>
<td>Updated for Abaqus 2020</td>
</tr>
<tr>
<td>Lesson 9</td>
<td>11/19</td>
<td>Updated for Abaqus 2020</td>
</tr>
<tr>
<td>Lesson 10</td>
<td>11/19</td>
<td>Updated for Abaqus 2020</td>
</tr>
<tr>
<td>Lesson 11</td>
<td>11/19</td>
<td>Updated for Abaqus 2020</td>
</tr>
<tr>
<td>Appendix 1</td>
<td>11/19</td>
<td>Updated for Abaqus 2020</td>
</tr>
<tr>
<td>Appendix 2</td>
<td>11/19</td>
<td>Updated for Abaqus 2020</td>
</tr>
<tr>
<td>Appendix 3</td>
<td>11/19</td>
<td>Updated for Abaqus 2020</td>
</tr>
<tr>
<td>Appendix 4</td>
<td>11/19</td>
<td>Updated for Abaqus 2020</td>
</tr>
<tr>
<td>Appendix 5</td>
<td>11/19</td>
<td>Updated for Abaqus 2020</td>
</tr>
<tr>
<td>Appendix 6</td>
<td>11/19</td>
<td>Updated for Abaqus 2020</td>
</tr>
<tr>
<td>Appendix 7</td>
<td>11/19</td>
<td>Updated for Abaqus 2020</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Workshop 1</th>
<th>11/19</th>
<th>Updated for Abaqus 2020</th>
</tr>
</thead>
<tbody>
<tr>
<td>Workshop 2a</td>
<td>11/19</td>
<td>Updated for Abaqus 2020</td>
</tr>
<tr>
<td>Workshop 2b</td>
<td>11/19</td>
<td>Updated for Abaqus 2020</td>
</tr>
<tr>
<td>Workshop 3</td>
<td>11/19</td>
<td>Updated for Abaqus 2020</td>
</tr>
<tr>
<td>Workshop 4</td>
<td>11/19</td>
<td>Updated for Abaqus 2020</td>
</tr>
<tr>
<td>Workshop 5</td>
<td>11/19</td>
<td>Updated for Abaqus 2020</td>
</tr>
<tr>
<td>Workshop 6</td>
<td>11/19</td>
<td>Updated for Abaqus 2020</td>
</tr>
<tr>
<td>Workshop 7</td>
<td>11/19</td>
<td>Updated for Abaqus 2020</td>
</tr>
<tr>
<td>Workshop 8</td>
<td>11/19</td>
<td>Updated for Abaqus 2020</td>
</tr>
<tr>
<td>Workshop 9</td>
<td>11/19</td>
<td>Updated for Abaqus 2020</td>
</tr>
<tr>
<td>Workshop 10</td>
<td>11/19</td>
<td>Updated for Abaqus 2020</td>
</tr>
</tbody>
</table>
Lesson 1: Introduction

Lesson content:

- Description of a Composite
- Some Typical Composites
- Finite Element Modeling of Composites

20 minutes
Lesson 2: Macroscopic Modeling

Lesson content:

- Introduction
- Anisotropic Elasticity
- Viscoelasticity
- Thermal Expansion
- Material Orientation
- Multiscale Modeling

45 minutes
Lesson 3: Laminate Modeling

Lesson content:

- Introduction
- Laminated Composite Shells
- Continuum Shell Elements
- Continuum Shell Meshing
- Continuum Solid Elements
- Continuum Solid Shell Elements
- Symmetry Conditions and Laminated Structures
- Workshop Preliminaries
- Workshop 1: The Pagano Plate Problem
Lesson 4: Composite Modeling with Abaqus

Lesson content:

- Introduction
- 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
- Workshop 2a: Buckling of a Laminate Panel
- Workshop 2b: Composite Wing Section
- Workshop 3: Composite Yacht Hull

3 hours
Lesson 5: Modeling Damage and Failure in Composites

Lesson content:

- Failure Criteria in Laminates
- Failure Theories
- Progressive Damage of Fiber-Reinforced Composites
- Example
- Import of Composite Damage Model
Lesson content:

- Introduction
- Cohesive Element Technology
- Constitutive Response in Cohesive Elements
- Viscous Regularization for Cohesive Elements
- Cohesive Element Examples
- Surface-based Cohesive Behavior
- Element-based vs. Surface-based Cohesive Behavior
- Workshop 4: Analysis of a DCB using Cohesive Behavior

Note: Appendix 2 contains an in-depth discussion of modeling techniques for cohesive elements using both the interactive and keywords interfaces.
Lesson content:

- Introduction
- VCCT Criterion
- LEFM Example using Abaqus/Standard
- LEFM Example using Abaqus/Explicit
- Output
- Ductile Fracture with VCCT
- VCCT Plug-in
- Comparison with Cohesive Behavior
- Examples
- Workshop 5: Analysis of a DCB using VCCT (Abaqus/Standard)
- Workshop 6: Analysis of a DCB using VCCT (Abaqus/Explicit)
Lesson 8: Reinforcement Modeling

**Lesson content:**

- Introduction
- Rebar Layers
- Embedded Elements

45 minutes
Lesson 9: Modeling of Sandwich Composites

Lesson content:

- Introduction to Sandwich Composites
- Abaqus Usage
- Modeling Skins with Abaqus/CAE
- Examples
  - Comparison to NAFEMS solution
  - Comparison of Conventional and Continuum Shells
  - Stacking Elements Through the Thickness
  - Tapered Sandwich Composite
- Workshop 7: Bending of a Sandwich Beam
Lesson 10: Modeling of Stiffened Panels

Lesson content:

- Stiffened Composite Panels
- Abaqus Usage
- Example
- Workshop 8: Bending of a Reinforced Flat Panel under Uniform Pressure

2 hours
Lesson 11: Fatigue Crack Growth at Material Interfaces

Lesson content:

- Introduction
- Direct Cyclic Procedure
- Linear Elastic Fatigue Crack Growth Analysis Procedure
- Fatigue Crack Growth Criterion
- Example: Fatigue Crack Growth Prediction for a DCB
- Workshop 9: Fatigue Crack Growth in a DCB Specimen

1 hour
Appendix 1: Debond Capability

Appendix content:

- Introduction
- Modeling Interface Behavior
Appendix 2: Cohesive Element Modeling Techniques

Appendix content:

- Viscous Regularization
- Modeling Techniques
Appendix 3: More on Continuum Shell Elements

Appendix content:

- Defining the Thickness Direction for Continuum Shell Elements
- Shell Thickness
- Change in Thickness and Thickness Modulus
Appendix 4: Alternative Modeling Techniques

*Appendix content:*

- Introduction
- Laminated Shell Section Definition
- Laminated Solid Section Definition
- Section Point-Based Postprocessing Technique
Appendix 5: Modeling Composite Material Impact

Appendix content:

- Introduction
- Composite Damage Models in Abaqus/Explicit
- Unidirectional Fiber
  - Example – Composite Plate Impact
- Woven Fabric
  - Example – Corrugated Beam Crushing
- Modeling Techniques
- Workshop 10: Perforation of a Composite Plate

1.5 hours
Appendix 6: Material Orientation Examples

Appendix content:

- Example 1: Layered Shell Elements
- Example 2: Solid Elements
- Example 3: Layered Solid Elements
Appendix 7: Multiscale Modeling

Appendix content:

- Introduction
- Mean-field Homogenization
- Mean-field Homogenization for Linear Elastic Composites
- Specifying the Microstructure of the Composite
- Validation: Unit Cube with Spherical Inclusion
- Validation: Matrix with Cylindrical Inclusion
- Fiber Orientation
- Example: Unidirectional stiffened panel subjected to axial compression
- Validation: Short Fiber Composites
- Multi-step Homogenization
- Example: Multiple Inclusion Model
- Composites with Thermal Expansion
- Incremental Mean-field Homogenization for Nonlinear Composites
- Output
- Examples
- Micromechanics Plug-in
- Upscaling
- Downscaling

1.5 hour