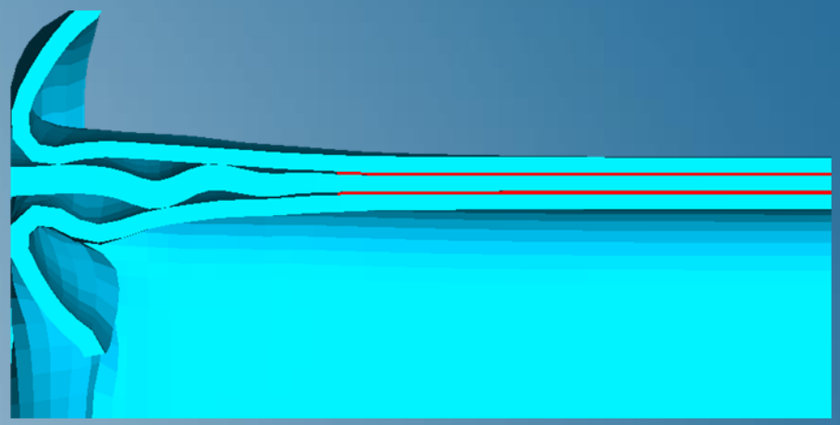
Analysis of Composite Materials with Abaqus

Abaqus 2022



3DEXPERIENCE®

About this Course

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

3 days

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

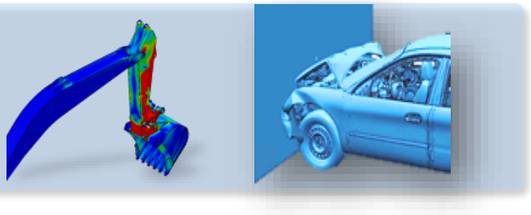
Additional Material

- 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

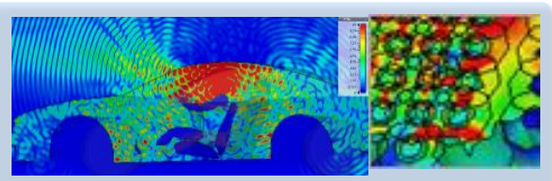
SIMULIA

- SIMULIA is the Dassault Systèmes brand for realistic simulation solutions.
- Advanced simulation portfolio covering simulation disciplines such as structural mechanics, computational fluid dynamics and electromagnetic field simulation, for a true multiphysics simulation approach.

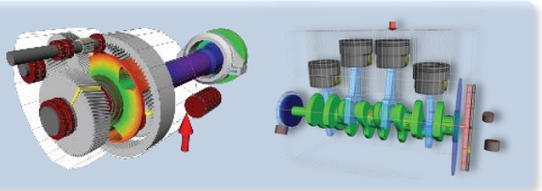
Structures



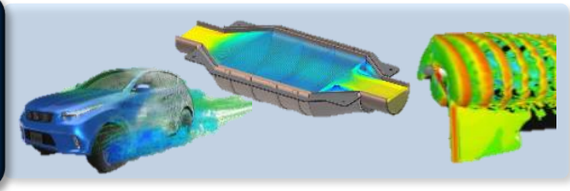
Electromagnetics



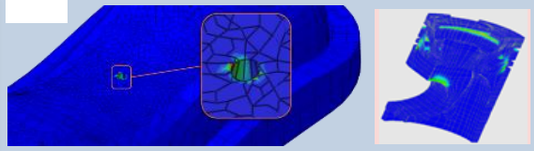
Multibody



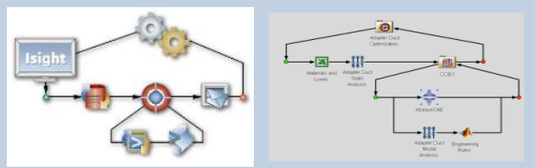
Fluids



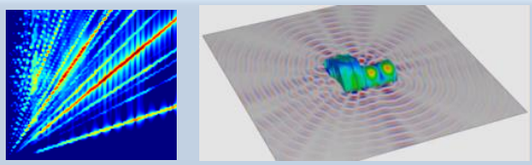
Durability



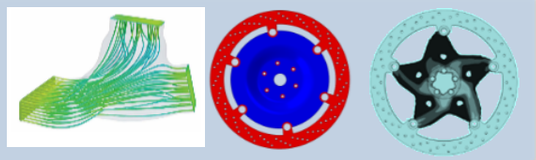
Automation



Vibro-acoustics



Optimization

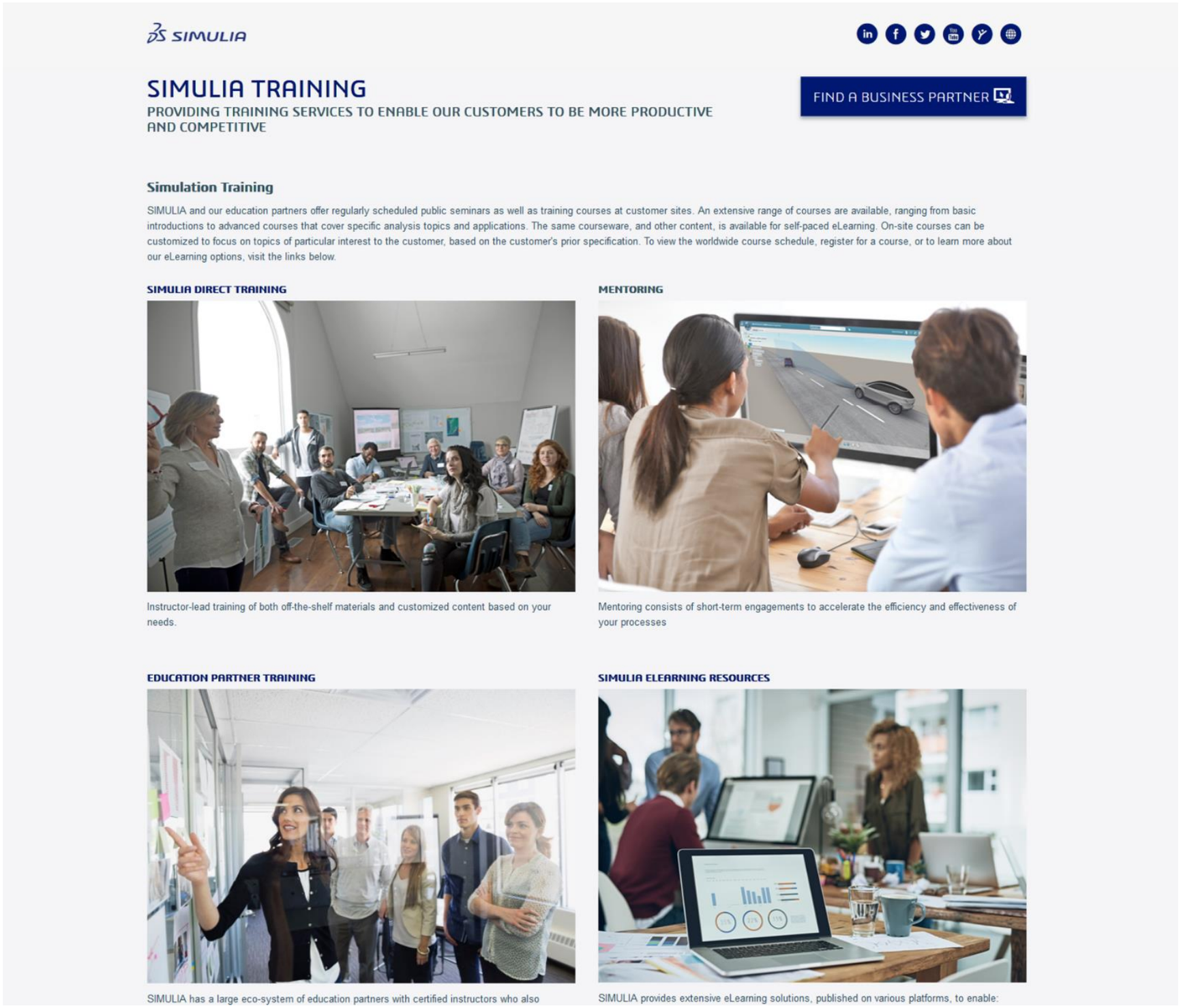


Join the Community!

Go to go.3ds.com/simc to log in or join!

SIMULIA Training

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



SIMULIA

SIMULIA TRAINING

PROVIDING TRAINING SERVICES TO ENABLE OUR CUSTOMERS TO BE MORE PRODUCTIVE AND COMPETITIVE

FIND A BUSINESS PARTNER

Simulation Training

SIMULIA and our education partners 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. The same courseware, and other content, is available for self-paced eLearning. 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, register for a course, or to learn more about our eLearning options, visit the links below.

SIMULIA DIRECT TRAINING

Instructor-lead training of both off-the-shelf materials and customized content based on your needs.

MENTORING

Mentoring consists of short-term engagements to accelerate the efficiency and effectiveness of your processes

EDUCATION PARTNER TRAINING

SIMULIA has a large eco-system of education partners with certified instructors who also

SIMULIA ELEARNING RESOURCES

SIMULIA provides extensive eLearning solutions, published on various platforms, to enable:

Legal Notices

Revision Status

Lesson 1	11/21	Updated for Abaqus 2022
Lesson 2	11/21	Updated for Abaqus 2022
Lesson 3	11/21	Updated for Abaqus 2022
Lesson 4	11/21	Updated for Abaqus 2022
Lesson 5	11/21	Updated for Abaqus 2022
Lesson 6	11/21	Updated for Abaqus 2022
Lesson 7	11/21	Updated for Abaqus 2022
Lesson 8	11/21	Updated for Abaqus 2022
Lesson 9	11/21	Updated for Abaqus 2022
Lesson 10	11/21	Updated for Abaqus 2022
Lesson 11	11/21	Updated for Abaqus 2022
Appendix 1	11/21	Updated for Abaqus 2022
Appendix 2	11/21	Updated for Abaqus 2022
Appendix 3	11/21	Updated for Abaqus 2022
Appendix 4	11/21	Updated for Abaqus 2022
Appendix 5	11/21	Updated for Abaqus 2022
Appendix 6	11/21	Updated for Abaqus 2022
Appendix 7	11/21	Updated for Abaqus 2022

Workshop 1	11/21	Updated for Abaqus 2022
Workshop 2a	11/21	Updated for Abaqus 2022
Workshop 2b	11/21	Updated for Abaqus 2022
Workshop 3	11/21	Updated for Abaqus 2022
Workshop 4	11/21	Updated for Abaqus 2022
Workshop 5	11/21	Updated for Abaqus 2022
Workshop 6	11/21	Updated for Abaqus 2022
Workshop 7	11/21	Updated for Abaqus 2022
Workshop 8	11/21	Updated for Abaqus 2022
Workshop 9	11/21	Updated for Abaqus 2022
Workshop 10	11/21	Updated for Abaqus 2022

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

3 hours

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

75 minutes

Lesson 6: Cohesive Behavior

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.

3 hours

Lesson 7: Virtual Crack Closure Technique (VCCT)

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
- Advanced Topics
  - Unstable crack growth
  - Smoothing crack tangential directions
- Workshop 5: Analysis of a DCB using VCCT (Abaqus/Standard)
- Workshop 6: Analysis of a DCB using VCCT (Abaqus/Explicit)

3 hours

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

1.5 hours

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
- Linear Elastic Fatigue Crack Growth Analysis Procedure
- Fatigue Crack Growth Criterion
- Example: Fatigue Crack Growth Prediction for a DCB
- Improving Crack Front Smoothness
- Workshop 9: Fatigue Crack Growth in a DCB Specimen

1 hour

Appendix 1: Debond Capability

Appendix content:

- Introduction
- Modeling Interface Behavior

30 minutes

Appendix 2: Cohesive Element Modeling Techniques

Appendix content:

- Viscous Regularization
- Modeling Techniques

1 hour

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

1 hour

Appendix 4: Alternative Modeling Techniques

Appendix content:

- Introduction
- Laminated Shell Section Definition
- Laminated Solid Section Definition
- Section Point-Based Postprocessing Technique

1 hour

Appendix 5: Modeling Composite Material Impact

Appendix content:

- Introduction
- Composite Damage Models in Abaqus/Explicit
- Unidirectional Fibers
  - Example – Composite Plate Impact
- Woven Fabrics
  - 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

30 minutes

Appendix 7: Multiscale Modeling

Appendix content:

- Introduction
- Linear vs. Nonlinear Short-fiber Reinforced Composites
- 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