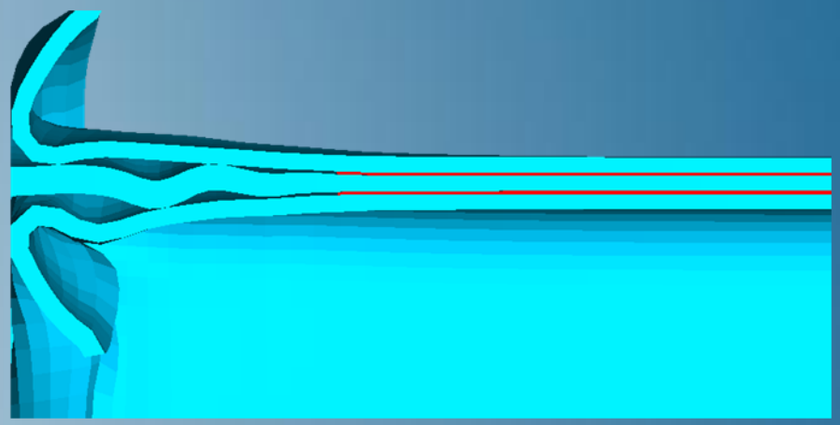
Analysis of Composite Materials with Abaqus

Abaqus 2018



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 low-cycle fatigue 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

- Lecture 1 Introduction
- Lecture 2 Macroscopic Modeling
- Lecture 3 Laminate Modeling
  - Workshop 1 The Pagano Plate Problem
- Lecture 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

- Lecture 5 Modeling Damage and Failure in Composites
- Lecture 6 Cohesive Behavior
  - Workshop 4 Analysis of a DCB using Cohesive Behavior
- Lecture 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

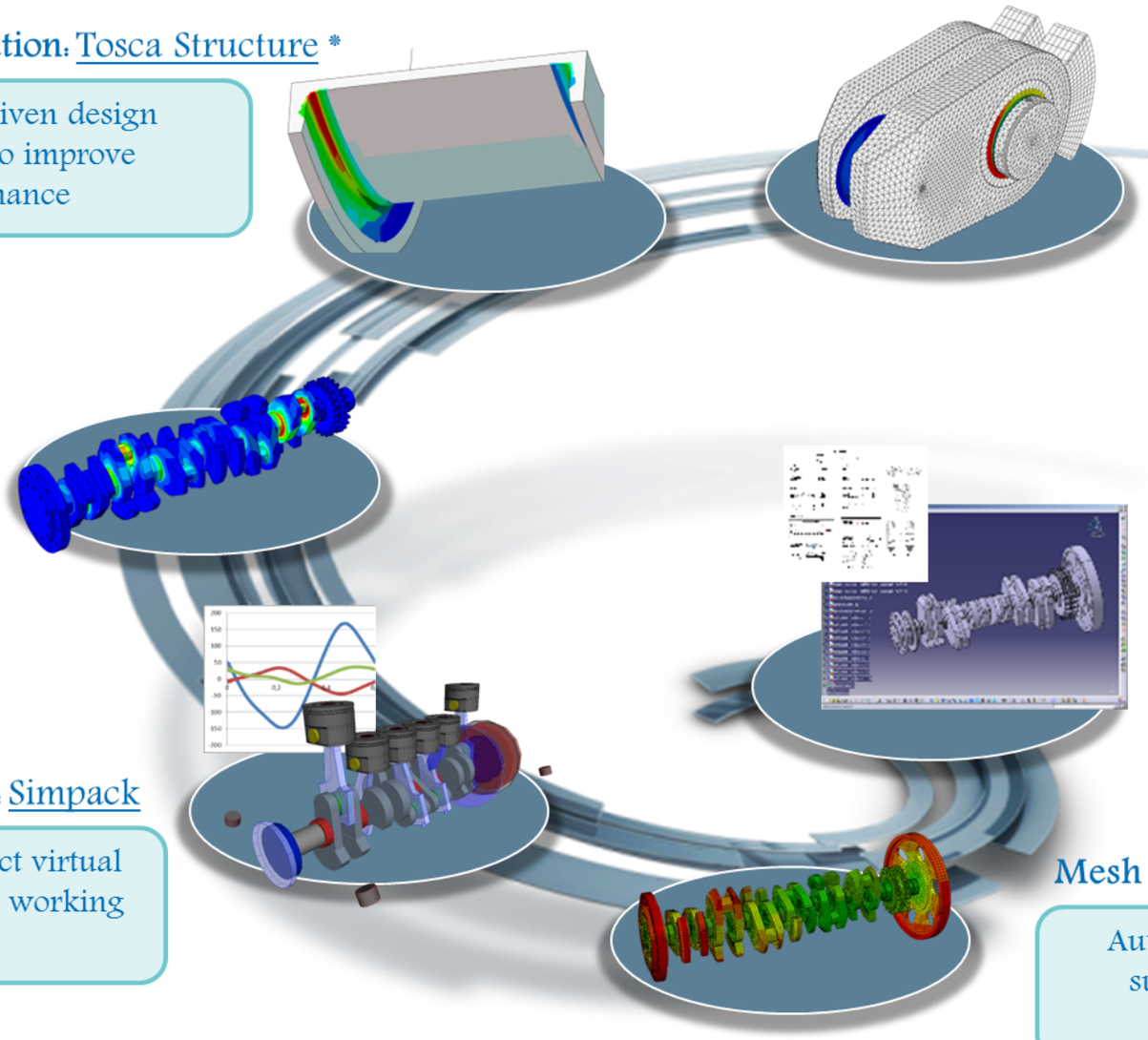
- Lecture 8 — Reinforcement Modeling
- Lecture 9 — Modeling of Sandwich Composites
  - Workshop 7 — Bending of a Sandwich Beam
- Lecture 10 — Modeling of Stiffened Panels
  - Workshop 8 — Bending of a Reinforced Flat Panel under Uniform Pressure
- Lecture 11 — Low-cycle Fatigue
  - Workshop 9 — Fatigue 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
- Portfolio of established, best-in-class products
  - Abaqus, Isight, Tosca, fe-safe, Simpack



Design Optimization: Tosca Structure *

Simulation-driven design refinement to improve performance

Durability Assessment: fe-safe *

Accurate life estimation to achieve certification

FEA Stress Analysis: Abaqus *

Detailed stress analysis using extracted load history from MBS

CAD Geometry: CATIA

Fully parameterized 3D geometry; FEA model generation via associative interface

Multibody Simulation: Simpack

System analysis to extract virtual load history of complete working cycle

Mesh Calibration: Isight *

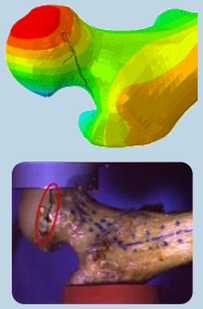
Automated mesh calibration: sufficient mesh quality for accurate results

* Included in extended licensing pool

SIMULIA's Power of the Portfolio

Abaqus

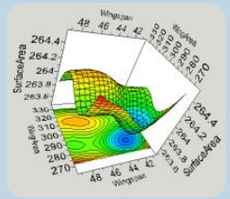
- Routine and Advanced Simulation
- Linear and Nonlinear, Static and Dynamic
- Thermal, Electrical, Acoustics
- Extended Physics through Co-simulation
- Model Preparation and Visualization



Realistic Human Simulation
High Speed Crash & Impact
Noise & Vibration

Isight

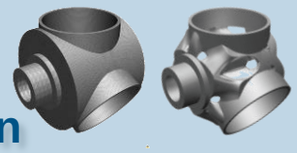
- Process Integration
- Design Optimization
- Parametric Optimization
- Six Sigma and Design of Experiments



Material Calibration
Workflow Automation
Design Exploration

Tosca

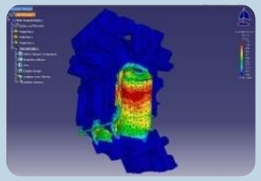
- Non-Parametric Optimization
- Structural and Fluid Flow Optimization
- Topology, Sizing, Shape, Bead Optimization



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

fe-safe

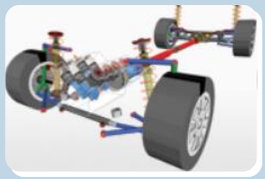
- Durability Simulation
- Low Cycle and High Cycle Fatigue
- Weld, High Temperature, Non-metallics



Safety Factors
Creep-Fatigue Interaction
Weld Fatigue

Simpack

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



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!

SIMULIA

Let the SIMULIA Learning Community be *Your* Portal to 21st Century Innovation

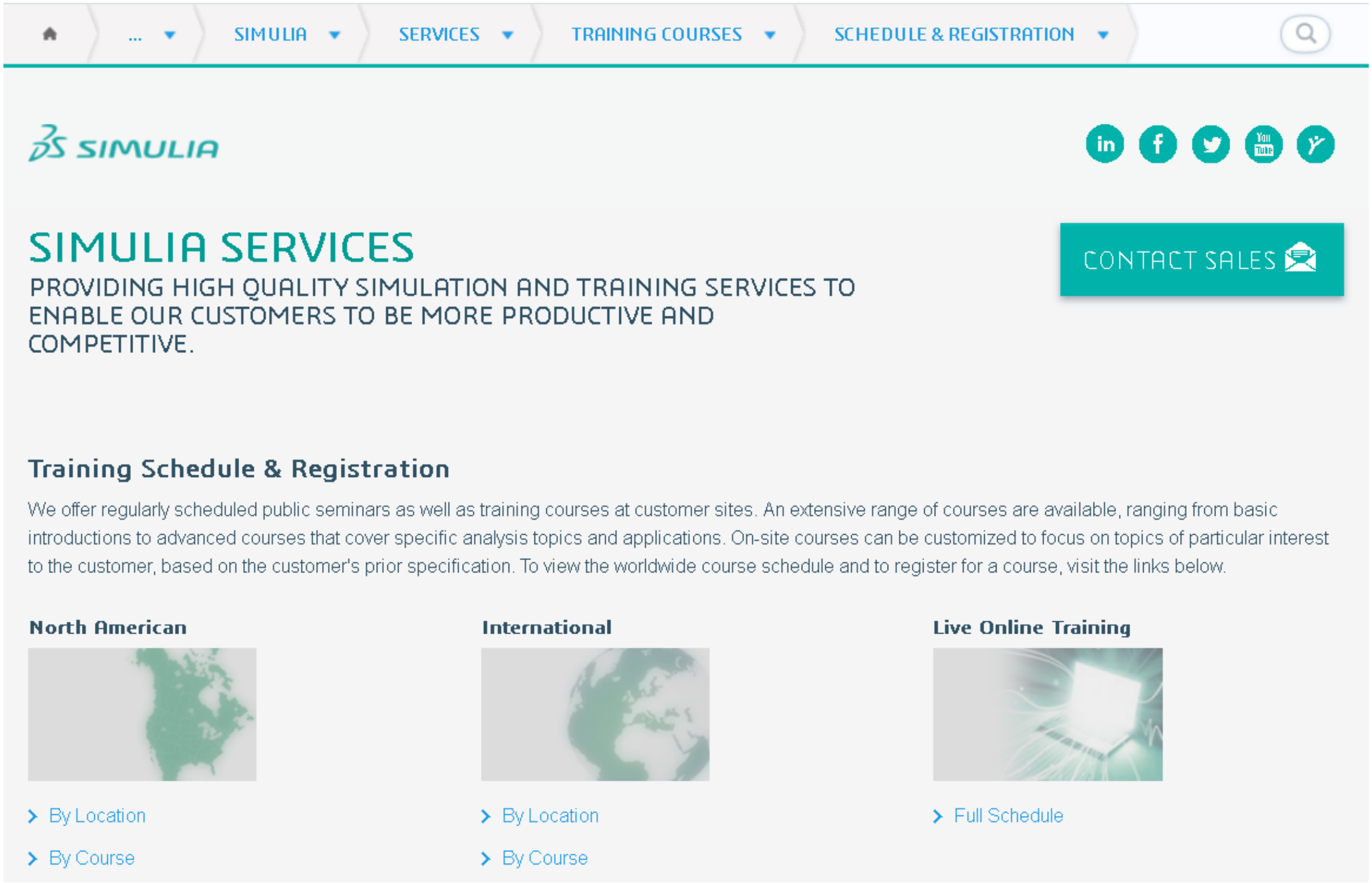
Discover new ways to explore how to leverage realistic simulation to drive product innovation. Join the thousands of Abaqus and Isight users who are already gaining valuable knowledge from the SIMULIA Learning Community.

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

DASSAULT SYSTEMES | The **3D**EXPERIENCE Company

SIMULIA Training

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



Legal Notices

Revision Status

Lecture 1	11/17	Updated for Abaqus 2018
Lecture 2	11/17	Updated for Abaqus 2018
Lecture 3	11/17	Updated for Abaqus 2018
Lecture 4	11/17	Updated for Abaqus 2018
Lecture 5	11/17	Updated for Abaqus 2018
Lecture 6	11/17	Updated for Abaqus 2018
Lecture 7	11/17	Updated for Abaqus 2018
Lecture 8	11/17	Updated for Abaqus 2018
Lecture 9	11/17	Updated for Abaqus 2018
Lecture 10	11/17	Updated for Abaqus 2018
Lecture 11	11/17	Updated for Abaqus 2018
Appendix 1	11/17	Updated for Abaqus 2018
Appendix 2	11/17	Updated for Abaqus 2018
Appendix 3	11/17	Updated for Abaqus 2018
Appendix 4	11/17	Updated for Abaqus 2018
Appendix 5	11/17	Updated for Abaqus 2018
Appendix 6	11/17	Updated for Abaqus 2018
Appendix 7	11/17	Updated for Abaqus 2018

Workshop 1	11/17	Updated for Abaqus 2018
Workshop 2a	11/17	Updated for Abaqus 2018
Workshop 2b	11/17	Updated for Abaqus 2018
Workshop 3	11/17	Updated for Abaqus 2018
Workshop 4	11/17	Updated for Abaqus 2018
Workshop 5	11/17	Updated for Abaqus 2018
Workshop 6	11/17	Updated for Abaqus 2018
Workshop 7	11/17	Updated for Abaqus 2018
Workshop 8	11/17	Updated for Abaqus 2018
Workshop 9	11/17	Updated for Abaqus 2018
Workshop 10	11/17	Updated for Abaqus 2018

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- 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
- 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: Low-cycle Fatigue

Lesson content:

- Introduction
- Direct Cyclic Low-cycle Fatigue Analysis
- Low-cycle Fatigue Criterion
- 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 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

30 minutes

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