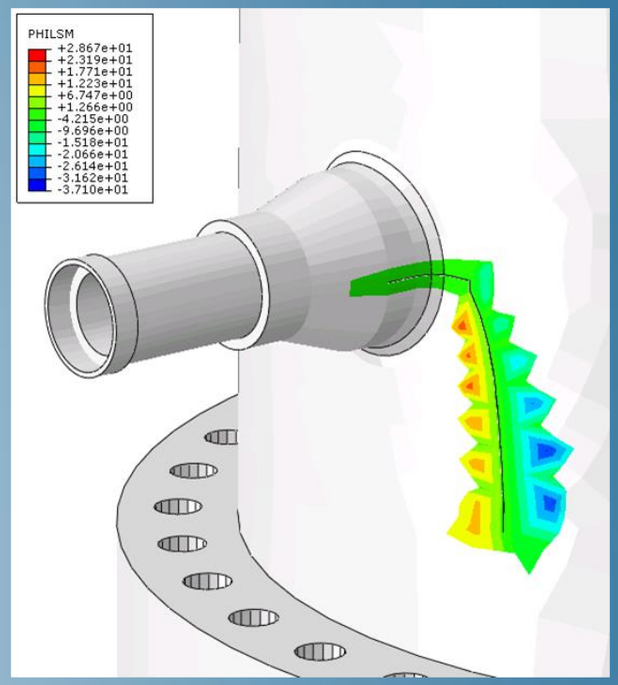
Modeling Fracture and Failure with Abaqus

Abaqus 2020



3DEXPERIENCE®

About this Course

Course objectives

Upon completion of this course you will be able to:

- Use proper modeling techniques to capture crack-tip singularities in fracture mechanics problems
- Use Abaqus/CAE to create meshes appropriate for fracture studies
- Calculate stress intensity factors and contour integrals around a crack tip
- Simulate material damage and failure
- Simulate crack growth using cohesive behavior, VCCT, and XFEM
- Simulate fatigue crack growth

Targeted audience

Simulation Analysts

Prerequisites

This course is recommended for engineers with experience using Abaqus

3 days

Day 1

- Lesson 1 — Basic Concepts of Fracture Mechanics
- Lesson 2 — Fracture Analysis of Sharp Cracks
  - Workshop 1 — Crack in a Three-point Bend Specimen
- Lesson 3 — General Fracture Analysis
  - Workshop 2 — Crack in a Helicopter Airframe Component

Day 2

- Lesson 4 Material Failure and Wear
- Lesson 5 Element-based Cohesive Behavior
  - Workshop 3 Crack Growth in a Three-point Bend Specimen using Cohesive Connections (Part 1)
  - Workshop 4 Crack Growth in a Helicopter Airframe Component using Cohesive Elements
- Lesson 6 Surface-based Cohesive Behavior
  - Workshop 3 Crack Growth in a Three-point Bend Specimen using Cohesive Connections (Part 2)

Day 3

- Lesson 7 Virtual Crack Closure Technology (VCCT)
  - Workshop 5 Crack Growth in a Three-point Bend Specimen using VCCT
- Lesson 8 Fatigue Crack Growth
- Lesson 9 Mesh-independent Fracture Modeling (XFEM)
  - Workshop 6 Crack Growth in a Three-point Bend Specimen using XFEM
  - Workshop 7 Modeling Crack Propagation in a Pressure Vessel with Abaqus using XFEM

Additional Material

- Appendix 1 — Other Fracture Mechanics Techniques
- Appendix 2 — Focused Mesh with Keywords

SIMULIA

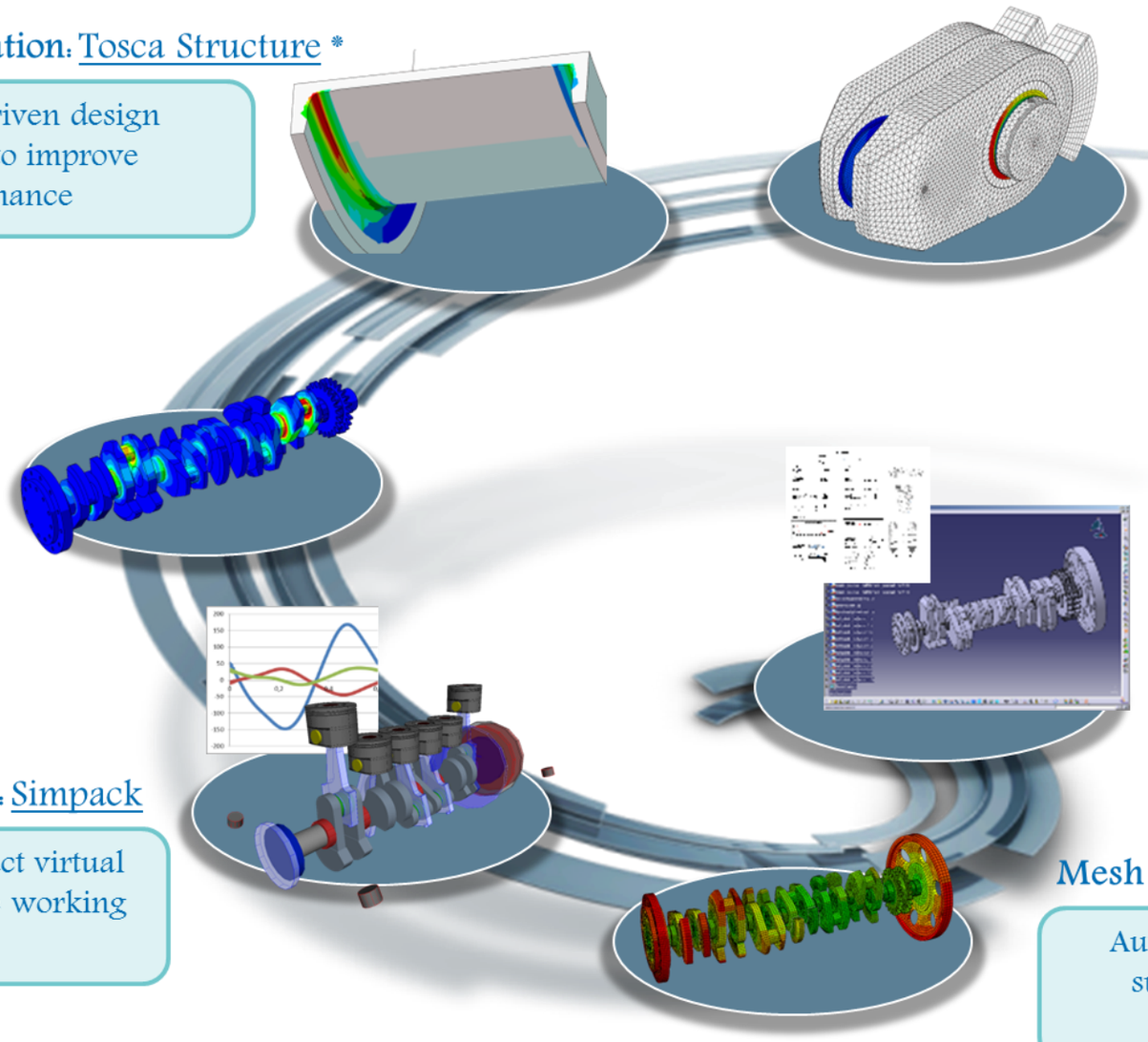
- SIMULIA is the Dassault Systèmes brand for Realistic Simulation solutions
- Portfolio of established, best-in-class products
  - Abaqus, Isight, Tosca, fe-safe, Simpack

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 *

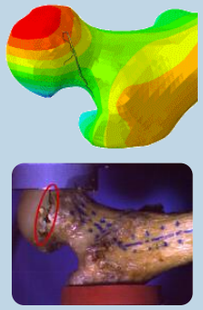
Automated mesh calibration: sufficient mesh quality for accurate results

* Included in extended licensing pool

SIMULIA's Power of the Portfolio

Abaqus

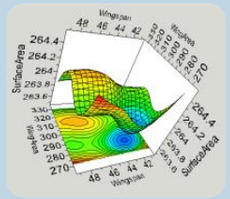
- Routine and Advanced Simulation
- Linear and Nonlinear, Static and Dynamic
- Thermal, Electrical, Acoustics
- Extended Physics through Co-simulation
- Model Preparation and Visualization



Realistic Human Simulation
High Speed Crash & Impact
Noise & Vibration

Isight

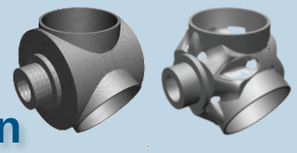
- Process Integration
- Design Optimization
- Parametric Optimization
- Six Sigma and Design of Experiments



Material Calibration
Workflow Automation
Design Exploration

Tosca

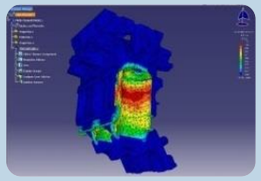
- Non-Parametric Optimization
- Structural and Fluid Flow Optimization
- Topology, Sizing, Shape, Bead Optimization



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

fe-safe

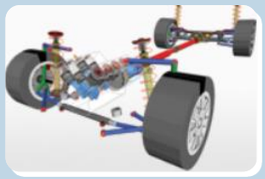
- Durability Simulation
- Low Cycle and High Cycle Fatigue
- Weld, High Temperature, Non-metallics



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

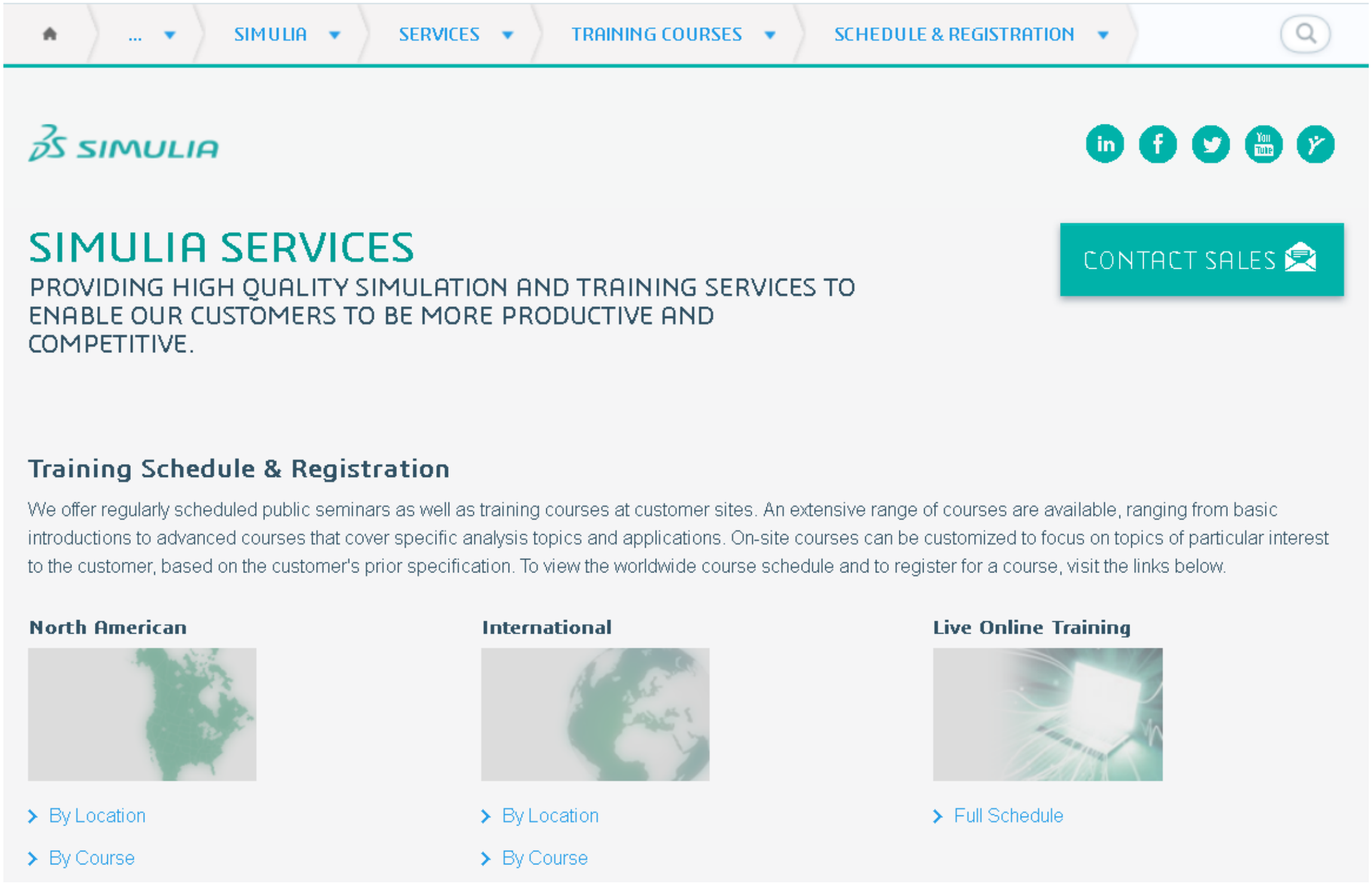
Discover new ways to explore how to leverage realistic simulation to drive product innovation. Join the thousands of Abaqus and Isight users who are already gaining valuable knowledge from the SIMULIA Learning Community.

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

Lesson 1	11/19	Updated for Abaqus 2020
Lesson 2	11/19	Updated for Abaqus 2020
Lesson 3	11/19	Updated for Abaqus 2020
Lesson 4	11/19	Updated for Abaqus 2020
Lesson 5	11/19	Updated for Abaqus 2020
Lesson 6	11/19	Updated for Abaqus 2020
Lesson 7	11/19	Updated for Abaqus 2020
Lesson 8	11/19	Updated for Abaqus 2020
Lesson 9	11/19	Updated for Abaqus 2020
Appendix 1	11/19	Updated for Abaqus 2020
Appendix 2	11/19	Updated for Abaqus 2020
Workshop 1	11/19	Updated for Abaqus 2020
Workshop 2	11/19	Updated for Abaqus 2020
Workshop 3	11/19	Updated for Abaqus 2020
Workshop 4	11/19	Updated for Abaqus 2020
Workshop 5	11/19	Updated for Abaqus 2020
Workshop 6	11/19	Updated for Abaqus 2020
Workshop 7	11/19	Updated for Abaqus 2020

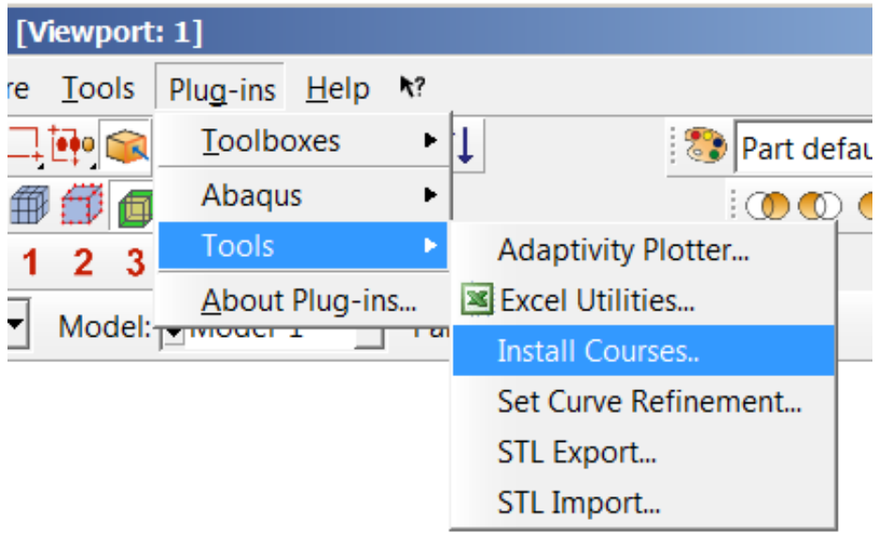
Environment Requirements for this course

This course does not contain any software installation files necessary to perform the exercises. In order to practice, you must have access to a software installation and environment that includes:

- Client application installed on your machine
  - Abaqus 2020

To install the files necessary to complete the workshop exercises, please do the following:

1. From the main menu bar of Abaqus/CAE, select **Plug-ins→Tools→Install Courses**.
2. In the **Install Courses** dialog box:
   - Specify the directory to which the files will be written.
   - Choose the course(s) for which the files will be extracted.
   - Click **OK**.



If you have any questions on how to access your environment, please contact your assigned Dassault Systèmes support team. You may also contact your education provider using the information on the **Contact us** page on the **Companion Learning Space** (**Help > Contact Us** menu.)

Lesson 1: Basic Concepts of Fracture Mechanics

Lesson content:

- Introduction
- Fracture Mechanisms
- Linear Elastic Fracture Mechanics
- Small Scale Yielding
- Energy Considerations
- The J-integral
- Mixed-Mode Fracture
- Fatigue
- Other Techniques

1 hour

Lesson 2: Fracture Analysis of Sharp Cracks

Lesson content:

- Crack Modeling Overview
- Modeling Sharp Cracks in Two Dimensions
- Modeling Sharp Cracks in Three Dimensions
- Calculation of Contour Integrals
- Examples
  - Penny-shaped crack in an infinite space
  - Conical crack in a half-space
  - Compact Tension Specimen
- Workshop Preliminaries
- Workshop 1: Crack in a Three-point Bend Specimen

2.5 hours

Lesson 3: General Fracture Analysis

Lesson content:

- Finite-Strain Analysis of Crack Tips
- Limitations of 3D Swept Meshing for Fracture
- Modeling Cracks with Keyword Options
- Nodal Normals in Contour Integral Calculations
- J-Integrals at Multiple Crack Tips
- Through Cracks in Shells
- Mixed-Mode Fracture
- Material Discontinuities
- Numerical Calculations with Elastic-Plastic Materials
- Residual Stresses
- Workshop 2: Crack in a Helicopter Airframe Component

2 hours

Lesson 4: Material Failure and Wear

Lesson content:

- Progressive Damage and Failure
- Damage Initiation Criteria for Ductile Metals
- Damage Evolution
- Element Removal
- Damage in Fiber-Reinforced Composite Materials
- Damage in Fasteners
- Material Wear and Ablation

2 hours

Lesson 5: Element-based Cohesive Behavior

Lesson content:

- Overview
- Introduction
- Element Technology
- Constitutive Response
- Viscous Regularization
- Modeling Techniques
- Examples
- Workshop 3: Crack Growth in a Three-point Bend Specimen using Cohesive Connections (Part 1)
- Workshop 4: Crack Growth in a Helicopter Airframe Component using Cohesive Elements

3 hours

Lesson 6: Surface-based Cohesive Behavior

Lesson content:

- Surface-based Cohesive Behavior
- Element-based vs. Surface-based Cohesive Behavior
- Workshop 3: Crack Growth in a Three-point Bend Specimen using Cohesive Connections (Part 2)

1.5 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: Crack Growth in a Three-point Bend Specimen using VCCT

2 hours

Lesson 8: Fatigue Crack Growth

Lesson content:

- Introduction
- Low-cycle Fatigue in Bulk Ductile Materials
- Linear Elastic Fatigue Crack Growth Analysis Procedure
- Fatigue Crack Growth Criterion
- Fatigue Crack Growth at Material Interfaces
- Fatigue Crack Growth in Bulk Brittle Materials
- Improving Crack Front Smoothness
- Summary

1 hour

Lesson 9: Mesh-independent Fracture Modeling (XFEM)

Lesson content:

- Introduction
- Basic XFEM Concepts
- Contact Modeling with XFEM
- Damage Modeling
- Cohesive Damage Modeling
- LEFM-based Damage Modeling
- Creating an XFEM Fracture Model
- Example 1 – Crack Initiation and Propagation using Cohesive Damage
- Example 2 – Crack Initiation and Propagation using LEFM
- Example 3 – Fatigue
- Example 4 – Propagation of an Existing Crack
- Example 5 – Delamination and Through-thickness Crack Propagation
- Example 6 – Contour Integrals
- Example 7 – Pressure Penetration
- Modeling Tips
- Limitations
- Workshop 6: Crack Growth in a Three-point Bend Specimen using XFEM
- Workshop 7: Modeling Crack Propagation in a Pressure Vessel with Abaqus using XFEM

3 hours

Appendix 1: Other Fracture Mechanics Techniques

Appendix content:

- Nonlinear Fracture Mechanics
- Creep Fracture
- Interfacial Fracture

0.5 hours

Appendix 2: Focused Mesh with Keywords

Appendix content:

- Generate a Focused Mesh with Keyword Options

15 minutes