Modeling Fracture and Failure with Abaqus

Abaqus 2020
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
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 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

## 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.
## Revision Status

<table>
<thead>
<tr>
<th>Component</th>
<th>Date</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Lesson 1</td>
<td>11/19</td>
<td>Updated for Abaqus 2020</td>
</tr>
<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>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>Workshop 1</td>
<td>11/19</td>
<td>Updated for Abaqus 2020</td>
</tr>
<tr>
<td>Workshop 2</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>
</tbody>
</table>
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
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
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)
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 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
Appendix content:

* Generate a Focused Mesh with Keyword Options*