Modeling Fracture and Failure with Abaqus
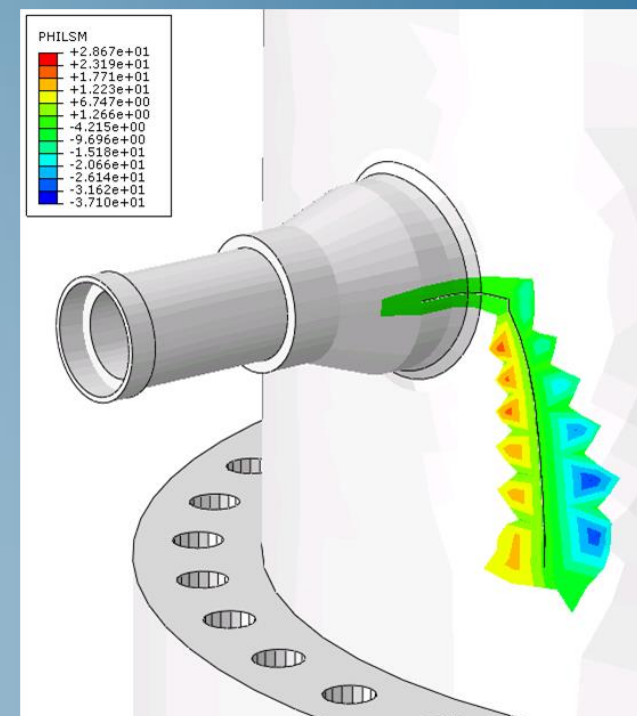
Abaqus 2018

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 low-cycle fatigue crack growth

Targeted audience
Simulation Analysts

Prerequisites
This course is recommended for engineers with experience using Abaqus

3 days

Day 1

▷ Lecture 1   Basic Concepts of Fracture Mechanics

▷ Lecture 2   Fracture Analysis of Sharp Cracks

      ◻ Workshop 1   Crack in a Three-point Bend Specimen

▷ Lecture 3   General Fracture Analysis

      ◻ Workshop 2   Crack in a Helicopter Airframe Component

Day 2

▷ **Lecture 4**   Material Failure and Wear

▷ **Lecture 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

▷ **Lecture 6**   Surface-based Cohesive Behavior

        ▢ **Workshop 3**   Crack Growth in a Three-point Bend Specimen using Cohesive Connections (Part 2)

Day 3

▷ Lecture 7   Virtual Crack Closure Technology (VCCT)

    ▫ Workshop 5   Crack Growth in a Three-point Bend Specimen using VCCT

▷ Lecture 8   Low-cycle Fatigue

▷ Lecture 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
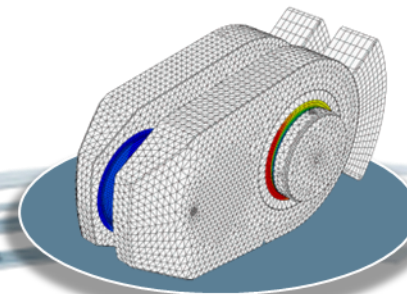 ▫ Abaqus, Isight, Tosca, fe-safe, Simpack

Design Optimization: Tosca Structure *

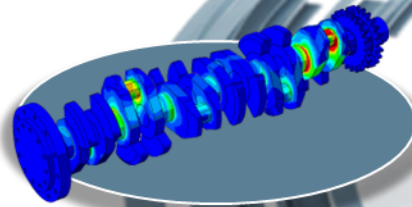Simulation-driven design refinement to improve performance
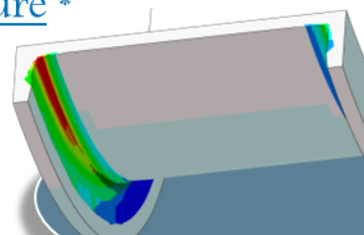
Durability Assessment: fe-safe *

Accurate life estimation to achieve certification

FEA Stress Analysis: Abaqus *
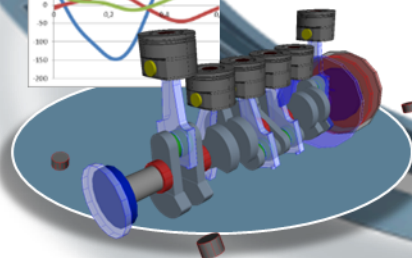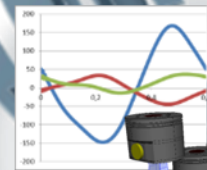
Detailed stress analysis using extracted load history from MBS

CAD Geometry: CATIA

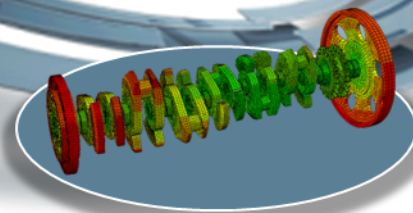Fully parameterized 3D geometry; FEA model generation via associative interface
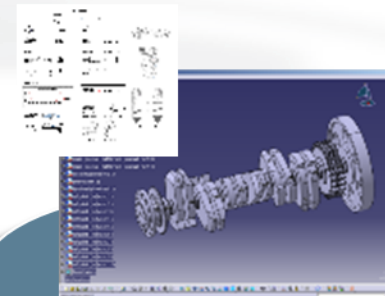
Multibody Simulation: Simpack

System analysis to extract virtual load history of complete working cycle

Mesh Calibration: Isight *

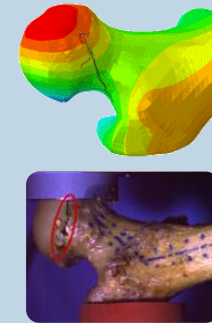Automated mesh calibration: sufficient mesh quality for accurate results

* 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
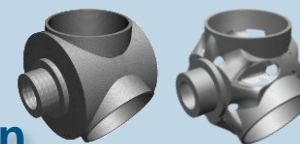Noise & Vibration**

Isight
- Process Integration
- Design Optimization
- Parametric Optimization
- Six Sigma and Design of Experiments
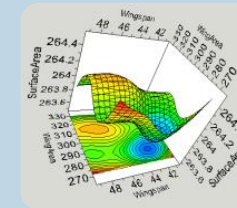
**Material Calibration
Workflow Automation
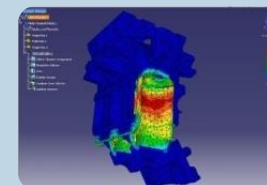Design Exploration**

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

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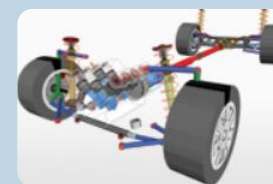Pressure Loss Reduction**

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!

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.

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

SIMULIA Training

Legal Notices

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.

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
Appendix 1	11/17	Updated for Abaqus 2018
Appendix 2	11/17	Updated for Abaqus 2018
Workshop 1	11/17	Updated for Abaqus 2018
Workshop 2	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

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- 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: Low-cycle Fatigue

Lesson content:

▷ Introduction

▷ Low-cycle Fatigue in Bulk Materials

▷ Low-cycle Fatigue at Material Interfaces

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 – Low Cycle 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