Learning Module

Modeling Fracture and Failure with Abaqus

Fracture and failure modeling allows for product designs that maximize the safe operating life of structural components. Abaqus offers many capabilities that enable fracture and failure modeling. This seminar provides a detailed discussion of these capabilities..

Objectives

Upon Completion Of This Course You Will Be Able To:
- Use proper modeling techniques for capturing 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.

Knowledge Prerequisites

This course is recommended for engineers with experience using Abaqus.

Brands

Simulia

Available Releases

SIMULIA 2021, SIMULIA 2020, SIMULIA 2019, SIMULIA 2018, SIMULIA 2017, SIMULIA 2016, SIMULIA V6.14, SIMULIA V6.13, SIMULIA V6.12

Duration

24 hours

Discipline

Advanced Abaqus

Language(s) for selected release

English

Contents