Learning Module

Introduction to fe-safe/Rubber

This 2-day course provides background information and hands-on experience for calculating fatigue of elastomers using Abaqus and fe-safe/Rubber.

Objectives

FEA Modeling for fe-safe/Rubber in Abaqus/CAE
Using the fe-safe/Rubber Interface.
Stress-strain relationship for rubber materials in Abaqus.
Theory of fatigue crack growth under relaxing and non-relaxing loads.
Overview of material calibration and how to enter material properties in fe-safe.
Calibrating the crack precursor size for fe-safe/Rubber.
Variable amplitude loading and multiple block loading for fe-safe/Rubber.
Postprocessing exports from fe-safe/Rubber (spreadsheet and Abaqus/Viewer).
Using fe-safe/Rubber output in the stand-alone Endurica damage sphere viewer.
Industry examples.

Knowledge Prerequisites

Introduction to Abaqus Introduction to fe-safe

Brands

Simulia

Available Releases

SIMULIA 2021, SIMULIA 2020, SIMULIA 2019, SIMULIA 2018

Duration

16 hours

Discipline

Fe-safe

Language(s) for selected release

English

Contents