Learning Module

Modeling Rubber and Viscoelasticity with Abaqus

This course provides a brief overview of finite deformations and the material models used for rubber and resilient foam.

Objectives

Upon Completion Of This Course You Will Be Able To:

- Use experimental test data to calculate material constants.
- Check the stability of the Abaqus material model at extreme strains.
- Obtain the best possible material constants from the available test data.
- Select elements for modeling rubber and foams.
- Design an appropriate finite element mesh.
- Model viscoelastic behavior in both the time and frequency domain.
- Use a user subroutine to define the hyperelastic behavior.

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

16 hours

Discipline

Advanced Abaqus

Language(s) for selected release

English

Contents