

## 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

Overview - Modeling Rubber and Viscoelasticity with Abaqus

- 1 - Rubber Physics
  - 2 - Introduction to Hyperelasticity Models
  - 3 - Mechanical Testing
  - 4 - Defining Rubber Elasticity Models in Abaqus
  - 5 - Modeling Issues and Tips
  - 6 - Viscoelastic Material Behavior
  - 7 - Time-Domain Viscoelasticity
  - 8 - Frequency-Domain Viscoelasticity
  - 9 - Permanent Set in Solid Elastomers
  - 10 - Anisotropic Hyperelasticity
- Appendices