

## Learning Module

# Tire Analysis with Abaqus: Fundamentals

---

Modern tires are among the most complex structures in production and their complexities span a broad range of the capabilities available in Abaqus. Since tire modeling is a specialized field, this seminar covers the many important yet basic capabilities in Abaqus that are specifically relevant to tire modeling.

### Objectives

In this course you will learn about:

- Choosing appropriate elements.
- Methods of modeling reinforcement.
- Contact modeling details pertinent to tire modeling.
- Fundamentals of material modeling-stress and strain measures, material directions.
- Linear elasticity, hyperelasticity and viscoelasticity.
- Efficient axisymmetric to three-dimensional model generation and results transfer.

### Knowledge Prerequisites

None

### Language(s) for selected release

English

### 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

## Contents

Overview - Tire Analysis with Abaqus - Fundamentals

- 1 - Tire Modeling Tools in Abaqus
- 2 - Axisymmetric Model Building
- 3 - Symmetric Model Generation and Results Transfer
- 4 - Three-dimensional Model Building
- 5 - Elements and Reinforcement
- 6 - Modeling Contact
- 7 - Rubber Models for Tire Analysis