

## Learning Module

# Modeling Contact and Resolving Convergence Issues with Abaqus

This 3-day course provides an in-depth discussion on solving nonlinear problems in Abaqus/Standard with an emphasis on modeling and convergence-related issues for contact. Engineers at Abaqus have developed many techniques and guidelines for solving challenging contact problems. Convergence issues related to complicated material models and geometrically unstable behavior are also covered. Many years of practical experience in understanding and resolving convergence issues have been condensed into this course.

### Objectives

Upon Completion Of This Course You Will Be Able To:

- Define general contact and contact pairs.
- Define appropriate surfaces (rigid or deformable).
- Model frictional contact.
- Model large sliding between deformable bodies.
- Resolve overclosures in interference fit problems.
- Understand how nonlinear problems are solved in Abaqus.
- Develop Abaqus models that will converge.
- Identify modeling errors that cause models to experience convergence difficulties.
- Recognize when a problem is too difficult or too illposed to be solved effectively.

### 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

### Duration

16 hours

### Discipline

Advanced Abaqus

### Language(s) for selected release

English

## Contents

Overview - Modeling Contact and Resolving Convergence Issues with Abaqus

- 1 - Introduction to Nonlinear FEA
  - 2 - Contact Workflow
  - 3 - Nonlinear FEA with Abaqus Standard
  - 4 - Why Abaqus Fails to Converge
  - 5 - Surface-based Contact
  - 6 - Solution of Unstable Problems
  - 7 - Contact Properties
  - 8 - Convergence Problems - Element Behavior
  - 9 - Convergence Problems – Materials
  - 10 - Interference Fits
  - 11 - Convergence Problems - Constraints and Loading
  - 12 - Modeling Tips
- Additional Material