

## Learning Module

# Element Selection in Abaqus

---

This course provides a brief overview of the distinguishing characteristics of the wide range of continuum and structural elements available in Abaqus for stress analyses. It explains modeling features that may cause certain types of elements to behave poorly.

### Objectives

Upon Completion Of This Course You Will Be Able To:

- Understand the distinguishing characteristics of the wide range of continuum and structural elements available in Abaqus for stress analyses.
- Understand modeling features that may cause certain types of elements to behave poorly.
- Choose appropriate element types for different applications including the effects of fully or nearly incompressible material behavior, contact, bending, etc.

### Knowledge Prerequisites

This course is recommended for engineers with experience using Abaqus

### 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 - Element Selection in Abaqus

- 1 - Solid Elements in Abaqus
- 2 - Other Solid Element Types
- 3 - Integration, Hourglassing, & Incompressibility
- 4 - Key Properties of Solid Elements
- 5 - Modeling Bending and Stress Concentrations
- 6 - Structural Elements in Abaqus
- 7 - Conventional Shell Elements
- 8 - Continuum Shell Elements
- 9 - Beam and Frame Elements