Learning Module

# Introduction to Abaqus

This course is a comprehensive and unified introduction to the modeling and analysis capabilities of Abaqus. It teaches you how to solve linear and nonlinear problems, submit and monitor analysis jobs and view simulation results using the interactive interface of Abaqus.

### Objectives

Upon completion of this course you will be able to:
- Use Abaqus/CAE to create complete finite element models.
- Use Abaqus/CAE to submit and monitor analysis jobs.
- Use Abaqus/CAE to view and evaluate simulation results.
- Solve structural analysis problems using Abaqus/Standard and Abaqus/Explicit, including the effects of material nonlinearity, large deformation and contact.

### Knowledge Prerequisites

None

### 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

32 hours

### Discipline

Advanced Abaqus

### Language(s) for selected release

English

## Contents

Overview - Introduction to Abaqus
1 - Overview of Abaqus
A First Look at Abaqus/CAE
2 - Working with Geometry (Part 1)
3 - Working with Geometry (Part 2)
4 - Material and Section Properties
5 - Assemblies in Abaqus
6 - Steps, Output, Loads, and Boundary Conditions
7 - Meshing Imported and Native Geometry
8 - Job Management and Results Visualization
9 - Linear and Nonlinear Problems
10 - Analysis Procedures (Part 1)
11 - Analysis Procedures (Part 2)
12 - Analysis Procedures (Part 3)
13 - Analysis Continuation Techniques
14 - Constraints and Connections
15 – Contact
Appendices

Общество с ограниченной ответственностью "Системы Инженерного Анализа"
603107, Россия, г. Н. Новгород, ул. Ларина, д. 7; ИНН 5261108537 / КПП 526101001
www.caecis.com, info@caecis.com, тел: +7 831 283 00 15