Learning Module

Introduction to Abaqus/Standard and Abaqus/Explicit

This introductory course is the ideal way to obtain a working knowledge of how to use both Abaqus/Standard and Abaqus/Explicit to solve linear and nonlinear problems. The seminar introduces you to the analysis capabilities of Abaqus using the keywords interface.

Objectives
Upon Completion Of This Course You Will Be Able To:
- Complete finite element models using Abaqus keywords.
- Submit and monitor analysis jobs.
- 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

Language(s) for selected release
English

Brands
Simulia

Available Releases
SIMULIA 2021, SIMULIA 2020, SIMULIA 2019, SIMULIA 2018, SIMULIA 2017, SIMULIA 2016, SIMULIA V6.12

Duration
24 hours

Discipline
Advanced Abaqus

Contents

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