Inceptra™

Hands on, Measurable Training Programs

Introduction to Abaqus

Course Code	EDU-SIMULIA 2018-ABI_F
Brand & Release	V 2018
Duration	4 days
Level	Fundamentals
Prerequisites	Basic knowledge of finite elements and continuum mechanics desirable. Also, some familiarity with interactive modeling is helpful but not required.

Objectives:

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. The following products will be covered: Abaqus/CAE, Abaqus Standard and Abaqus Explicit.

Class Structure:

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

Class Lessons:

Lesson 1: Overview of Abaqus
Lesson 2: Working with Geometry in Abaqus
Lesson 3: Working with Models created outside Abaqus
Lesson 4: Material and Section Properties
Lesson 5: Assemblies in Abaqus
Lesson 6: Steps, Output, Loads, and Boundary Conditions
Lesson 7: Meshing Imported and Native Geometry
Lesson 8: Job Management and Results Visualization
Lesson 9: Linear and Nonlinear Problems
Lesson 10: Static, General Analysis procedure
Lesson 11: Static, Linear Perturbation; Buckle & Frequency procedures; Multistep Analyses
Lesson 12: Dynamic, Explicit procedure
Lesson 13: Analysis Continuation Techniques
Lesson 14: Constraints and Connections
Lesson 15: Contact
Appendix: Element Selection Criteria; Analyzing Nonlinear Quasi-Static Problems; Heat Transfer & Thermal-Stress Analysis

©Inceptra All Rights Reserved.