Inceptra

Hands On, Measurable Training Programs

Introduction to Abaqus

Course Code	EDU-SIMULIA 2020-ABI_F
Brand & Release	SIMULIA 2020
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/monitor analysis jobs and view/evaluate simulation results
- Solve structural analysis problems using Abaqus/Standard and Abaqus/Explicit, including the effects of material nonlinearity, large deformation and contact

The course covers the following topics:

- Linear and nonlinear structural analysis
- Static, dynamic and heat transfer analysis
- Material models: linear elasticity, hyperelasticity and metal plasticity.
- Loads and constraints
- Modeling contact
- Selecting the appropriate elements for your problem
- Feature-based modeling, parts and assemblies
- Working with CAD geometry and imported meshes
- Mesh generation techniques
- Restarting an analysis

Class Lessons:

Lesson 1: Overview of Abaqus
Lesson 2: Working with Geometry (Part 1)
Lesson 3: Working with Geometry (Part 2)
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: Analysis Procedure (Part 1)
Lesson 11: Analysis Procedure (Part 2)
Lesson 12: Analysis Procedure (Part 3)
Lesson 13: Analysis Continuation Techniques
Lesson 14: Constraints and Connections
Lesson 15: Contact