Inceptra

Hands On, Measurable Training Programs

Introduction to Abaqus/CAE

Course Code	EDU-SIMULIA 2020-ICAE_F
Brand & Release	SIMULIA 2020
Duration	2 days
Level	Fundamentals
Prerequisites	Intended for new Abaqus/CAE users. Some familiarity with interactive preprocessors is helpful but not required.

Objectives:

Abaqus/CAE provides a complete interactive environment for creating Abaqus models, submitting and monitoring analysis jobs and viewing and manipulating simulation results. This course is intended to complement the Introduction to Abaqus/Standard and Abaqus/Explicit course, which describes how to perform linear and nonlinear analyses with Abaqus.

Class Structure:

The course offers an overview of the important features available in Abaqus/CAE:

- Creating parts using the feature-based modeler
- Importing parts into Abaqus/CAE
- Partitioning parts
- Meshing
- Defining analysis attributes
- Submitting and managing Abaqus simulations
- Viewing the results of the simulations

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

Class Lessons:

Lesson 1: Introducing Abaqus/CAE
Lesson 2: Working with Geometry in Abaqus/CAE
Lesson 3: Working with models created outside Abaqus
Lesson 4: Material Properties and Assemblies
Lesson 5: Steps, Interactions, and Loads
Lesson 6: Meshing
Lesson 7: Job Management and Results Visualization