SIMULIA Introduction to ABAQUS

OBJECTIVE: 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 are covered by this seminar: Abaqus/CAE, Abaqus/Standard and Abaqus/Explicit. This course is divided into lectures, demonstrations and workshops. The course's workshops are integral to the training. They are designed to reinforce concepts presented during the lectures and demonstrations. They are intended to provide users with the experience of running and trouble-shooting actual Abaqus analyses.

DURATION	STUDENT PROFILE	PRE-REQUISITES
4 Days	Simulation & Abaqus Users	None

TOPIC	DETAILS	DURATION
Introduction to Abaqus	▪ 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 ▪ Creating, submitting and monitoring analysis jobs ▪ Viewing simulation results ▪ Restarting an analysis	4 DAYS