Introduction to Abaqus (ABI)

Course Code	SIM-en-ABI-F-V30R2016
Available Release	2016
Duration	40 hours
Course Material	English
Level	Fundamental
Audience	Simulation Analysts
Description	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.
Objectives	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.
Prerequisites	None
Available Online	Yes