

Training Course

# MECANICA SOLUTIONS 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 4 DAYS TRAINING + 1 DAY MENTORING WORKSHOP**

**STUDENT PROFILE: SIMULATION/ABAQUS USERS**

TOPIC & DETAILS	TOPIC DURATION
<p><b>Introduction to Abaqus</b></p> <ul style="list-style-type: none"><li>• Linear and nonlinear structural analysis</li><li>• Static, dynamic and heat transfer analysis</li><li>• Material models: linear elasticity, hyperelasticity, and metal plasticity.</li><li>• Loads and constraints</li><li>• Modeling contact</li><li>• Selecting the appropriate elements for your problem</li><li>• Feature-based modeling, parts and assemblies</li><li>• Working with CAD geometry and imported meshes</li><li>• Mesh generation techniques</li><li>• Creating, submitting and monitoring analysis jobs</li><li>• Viewing simulation results</li><li>• Restarting an analysis</li></ul>	4 Days



Register on-line or call 1-888-326-8326

Information contained within is subject to change.

All classes are dependent on minimum enrollment

[www.mecanicasolutions.com](http://www.mecanicasolutions.com)

