Introduction to Abaqus/Standard and Abaqus/Explicit
Course objectives
Upon completion of this course you will be able to:

- Complete finite element models using Abaqus keywords.
- Submit and monitor analysis jobs.
- 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.

Targeted audience
Simulation Analysts

Prerequisites
None
Day 1

Lesson 1   Defining an Abaqus Model

Workshop 1 Basic Input and Output

Lesson 2   Linear Static Analysis

Workshop 2 Linear Static Analysis of a Cantilever Beam: Multiple Load Cases

Lesson 3   Nonlinear Analysis in Abaqus/Standard

Workshop 3 Nonlinear Statics
Day 2

Lesson 4  Multistep Analysis in Abaqus

Workshop 4  Unloading Analysis

Lesson 5  Constraints and Contact

Workshop 5  Seal Contact

Lesson 6  Introduction to Dynamics

Workshop 6  Dynamics
<table>
<thead>
<tr>
<th>Day 3</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Lesson 7</strong></td>
</tr>
<tr>
<td><strong>Workshop 7</strong></td>
</tr>
<tr>
<td><strong>Lesson 8</strong></td>
</tr>
<tr>
<td><strong>Workshop 8</strong></td>
</tr>
<tr>
<td><strong>Lesson 9</strong></td>
</tr>
<tr>
<td><strong>Workshop 9</strong></td>
</tr>
</tbody>
</table>
## Additional Material

<table>
<thead>
<tr>
<th>Appendix 1</th>
<th>Element Selection Criteria</th>
</tr>
</thead>
<tbody>
<tr>
<td>Appendix 2</td>
<td>Contact Issues Specific to Abaqus/Standard</td>
</tr>
<tr>
<td>Appendix 3</td>
<td>Contact Issues Specific to Abaqus/Explicit</td>
</tr>
</tbody>
</table>
Join the Community!

How can you maximize the robust technology of Abaqus FEA and Isight?
Connect with peers to share knowledge and get technical insights

Let the SIMULIA Learning Community be Your Portal to 21st Century Innovation
Discover new ways to explore how to leverage realistic simulation to drive product innovation. Join the thousands of Abaqus and Isight users who are already gaining valuable knowledge from the SIMULIA Learning Community.

For more information and registration, visit 3ds.com/simulia-learning.

©2013 Dassault Systèmes. All rights reserved.
The Abaqus Software described in this documentation is available only under license from Dassault Systèmes or its subsidiary and may be used or reproduced only in accordance with the terms of such license.

This documentation and the software described in this documentation are subject to change without prior notice.

Dassault Systèmes and its subsidiaries shall not be responsible for the consequences of any errors or omissions that may appear in this documentation.

No part of this documentation may be reproduced or distributed in any form without prior written permission of Dassault Systèmes or its subsidiary.

© Dassault Systèmes, 2014

Printed in the United States of America.

Abaqus, the 3DS logo, SIMULIA, and CATIA are trademarks or registered trademarks of Dassault Systèmes or its subsidiaries in the US and/or other countries.

Other company, product, and service names may be trademarks or service marks of their respective owners. For additional information concerning trademarks, copyrights, and licenses, see the Legal Notices in the Abaqus 6.14 Installation and Licensing Guide.
<table>
<thead>
<tr>
<th>Lesson 1</th>
<th>5/14</th>
<th>Updated for 6.14</th>
</tr>
</thead>
<tbody>
<tr>
<td>Lesson 2</td>
<td>5/14</td>
<td>Updated for 6.14</td>
</tr>
<tr>
<td>Lesson 3</td>
<td>5/14</td>
<td>Updated for 6.14</td>
</tr>
<tr>
<td>Lesson 4</td>
<td>5/14</td>
<td>Updated for 6.14</td>
</tr>
<tr>
<td>Lesson 5</td>
<td>5/14</td>
<td>Updated for 6.14</td>
</tr>
<tr>
<td>Lesson 6</td>
<td>5/14</td>
<td>Updated for 6.14</td>
</tr>
<tr>
<td>Lesson 7</td>
<td>5/14</td>
<td>Updated for 6.14</td>
</tr>
<tr>
<td>Lesson 8</td>
<td>5/14</td>
<td>Updated for 6.14</td>
</tr>
<tr>
<td>Lesson 9</td>
<td>5/14</td>
<td>Updated for 6.14</td>
</tr>
<tr>
<td>Appendix 1</td>
<td>5/14</td>
<td>Updated for 6.14</td>
</tr>
<tr>
<td>Appendix 2</td>
<td>5/14</td>
<td>Updated for 6.14</td>
</tr>
<tr>
<td>Appendix 3</td>
<td>5/14</td>
<td>Updated for 6.14</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Workshop 1</th>
<th>5/14</th>
<th>Updated for 6.14</th>
</tr>
</thead>
<tbody>
<tr>
<td>Workshop 2</td>
<td>5/14</td>
<td>Updated for 6.14</td>
</tr>
<tr>
<td>Workshop 3</td>
<td>5/14</td>
<td>Updated for 6.14</td>
</tr>
<tr>
<td>Workshop 4</td>
<td>5/14</td>
<td>Updated for 6.14</td>
</tr>
<tr>
<td>Workshop 5</td>
<td>5/14</td>
<td>Updated for 6.14</td>
</tr>
<tr>
<td>Workshop 6</td>
<td>5/14</td>
<td>Updated for 6.14</td>
</tr>
<tr>
<td>Workshop 7</td>
<td>5/14</td>
<td>Updated for 6.14</td>
</tr>
<tr>
<td>Workshop 8</td>
<td>5/14</td>
<td>Updated for 6.14</td>
</tr>
<tr>
<td>Workshop 9</td>
<td>5/14</td>
<td>Updated for 6.14</td>
</tr>
</tbody>
</table>
Lesson 1: Defining an Abaqus Model

Lesson content:

- Introduction
- Documentation
- Learning Community
- Components of an Abaqus Model
- Details of an Abaqus Input File
- Abaqus Input Conventions
- Abaqus Output
- Example: Cantilever Beam Model
- Parts and Assemblies (optional)
- Workshop Preliminaries
- Workshop 1: Basic Input and Output (IA)
- Workshop 1: Basic Input and Output (KW)

Both interactive (IA) and keywords (KW) versions of the workshop are provided. Complete only one.
Lesson 2: Linear Static Analysis

Lesson content:

- Linear and Nonlinear Procedures
- Linear Static Analysis and Multiple Load Cases
- Multiple Load Case Usage
- Examples
- Workshop 2: Linear Static Analysis of a Cantilever Beam (IA)
- Workshop 2: Linear Static Analysis of a Cantilever Beam (KW)

Both interactive (IA) and keywords (KW) versions of the workshop are provided. Complete only one.
Lesson content:

- Nonlinearity in Structural Mechanics
- Equations of Motion
- Nonlinear Analysis Using Implicit Methods
- Nonlinear Analysis Using Explicit Methods
- Input File for Nonlinear Analysis
- Status File
- Message File
- Output from Nonlinear Cantilever Beam Analysis
- Workshop 3: Nonlinear Statics (IA)
- Workshop 3: Nonlinear Statics (KW)

Both interactive (IA) and keywords (KW) versions of the workshop are provided. Complete only one.
Lesson 4: Multistep Analysis in Abaqus

Lesson content:

- Multistep Analyses
- Restart Analysis in Abaqus
- Workshop 4: Unloading Analysis (IA)
- Workshop 4: Unloading Analysis (KW)

Both interactive (IA) and keywords (KW) versions of the workshop are provided. Complete only one.
Lesson content:

- Constraints
- Tie Constraints
- Rigid Bodies
- Shell-to-solid Coupling
- Contact
- Defining General Contact
- Defining Contact Pairs
- Contact Pair Surfaces
- Local Surface Behavior
- Relative Sliding of Points in Contact
- Adjusting Initial Nodal Locations for Contact
- Contact Output
- Workshop 5: Seal Contact (IA)
- Workshop 5: Seal Contact (KW)

Both interactive (IA) and keywords (KW) versions of the workshop are provided. Complete only one.

2.5 hours
Lesson 6: Introduction to Dynamics

Lesson content:

- What Makes a Problem Dynamic?
- Equations for Dynamic Problems
- Linear Dynamics
- Nonlinear Dynamics
- Comparing Abaqus/Standard and Abaqus/Explicit
- Nonlinear Dynamics Example
- Workshop 6: Dynamics (IA)
- Workshop 6: Dynamics (KW)

Both interactive (IA) and keywords (KW) versions of the workshop are provided. Complete only one.
Lesson content:

- Overview of the Explicit Dynamics Procedure
- Abaqus/Explicit Syntax
- Rigid Bodies
- Workshop 7: Contact with Abaqus/Explicit (IA)
- Workshop 7: Contact with Abaqus/Explicit (KW)

Both interactive (IA) and keywords (KW) versions of the workshop are provided. Complete only one.
Lesson content:

- Introduction
- Solution Strategies
- Quasi-Static Simulations Using Explicit Dynamics
- Energy Balance
- Example: Load Rates
- Example: Mass Scaling
- Adaptive Meshing
- Workshop 8: Quasi-Static Analysis (IA)
- Workshop 8: Quasi-Static Analysis (KW)

Both interactive (IA) and keywords (KW) versions of the workshop are provided. Complete only one.
Lesson 9: Combining Abaqus/Standard & Abaqus/Explicit

Lesson content:

- Introduction
- Abaqus Usage
- Springback Calculation using Abaqus/Standard
- Workshop 9: Import Analysis (IA)
- Workshop 9: Import Analysis (KW)

Both interactive (IA) and keywords (KW) versions of the workshop are provided. Complete only one.
Appendix 1: Element Selection Criteria

Appendix content:

- Elements
- Structural (Shells and Beams) vs. Continuum Elements
- Modeling Bending Using Continuum Elements
- Stress Concentrations
- Contact
- Incompressible Materials
- Mesh Generation
- Solid Element Selection Summary
Appendix 2: Contact Issues Specific to Abaqus/Standard

Appendix content:

- Contact as Part of the Model Definition
- Mesh Density Considerations
- Contact Logic in Abaqus/Standard
Appendix 3: Contact Issues Specific to Abaqus/Explicit

Appendix content:

- Contact Pairs as Part of the History Data
- Enforcing the Contact Constraints
- Double-Sided Contact
- Initial Kinematic Compliance