Abaqus for Offshore Analysis
2017
Course objectives
The topics covered in this course include:

- Review nonlinear material behavior (metal plasticity and hyperelasticity)
- Capabilities of Abaqus element types in general
- Specific element discussions include drag chain, pipe, PSI and ITT elements
- Pipe-soil interaction, including lateral buckling of a pipe line on a seabed
- Abaqus/Aqua capabilities in Abaqus/Standard to model wave, buoyancy, current & wind loading
- Coupled Eulerian-Lagrangian (CEL) approach in Abaqus/Explicit

Targeted audience
This course is recommended for engineers with experience using Abaqus who work in the Oil and Gas industry.

Prerequisites
None
Day 2

- Lecture 7  
  Special Purpose Elements (Part 2)

- Lecture 8  
  Seabed Pipe Interaction

  - Workshop 5  
    Lateral Buckling of a Sub-Sea Pipeline
  
  - Workshop 6  
    Threaded Connector Analysis

- Lecture 9  
  Abaqus/Aqua Capabilities

  - Workshop 7  
    Abaqus/Aqua Example

- Lecture 10  
  Coupled Eulerian-Lagrangian (CEL) Approach

  - Demo 3  
    CEL interface in Abaqus/CAE
  
  - Workshop 8  
    Deformation of an Elastic Dam under Time-dependent Water Pressure

- Lecture 11  
  Modeling Tips & Special Analysis Techniques
SIMULIA is the Dassault Systèmes brand for Realistic Simulation solutions

- Portfolio of established, best-in-class products
  - Abaqus, Isight, Tosca, fe-safe, Simpack

* Included in extended licensing pool
<table>
<thead>
<tr>
<th>SIMULIA’s Power of the Portfolio</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Abaqus</strong></td>
</tr>
<tr>
<td>- Routine and Advanced Simulation</td>
</tr>
<tr>
<td>- Linear and Nonlinear, Static and Dynamic</td>
</tr>
<tr>
<td>- Thermal, Electrical, Acoustics</td>
</tr>
<tr>
<td>- Extended Physics through Co-simulation</td>
</tr>
<tr>
<td>- Model Preparation and Visualization</td>
</tr>
<tr>
<td><strong>Realistic Human Simulation</strong></td>
</tr>
<tr>
<td>High Speed Crash &amp; Impact Noise &amp; Vibration</td>
</tr>
<tr>
<td><strong>Isight</strong></td>
</tr>
<tr>
<td>- Process Integration</td>
</tr>
<tr>
<td>- Design Optimization</td>
</tr>
<tr>
<td>- Parametric Optimization</td>
</tr>
<tr>
<td>- Six Sigma and Design of Experiments</td>
</tr>
<tr>
<td><strong>Material Calibration</strong></td>
</tr>
<tr>
<td>Workflow Automation</td>
</tr>
<tr>
<td>Design Exploration</td>
</tr>
<tr>
<td><strong>Tosca</strong></td>
</tr>
<tr>
<td>- Non-Parametric Optimization</td>
</tr>
<tr>
<td>- Structural and Fluid Flow Optimization</td>
</tr>
<tr>
<td>- Topology, Sizing, Shape, Bead Optimization</td>
</tr>
<tr>
<td><strong>Conceptual/Detailed Design</strong></td>
</tr>
<tr>
<td>Weight, Stiffness, Stress</td>
</tr>
<tr>
<td>Pressure Loss Reduction</td>
</tr>
<tr>
<td><strong>fe-safe</strong></td>
</tr>
<tr>
<td>- Durability Simulation</td>
</tr>
<tr>
<td>- Low Cycle and High Cycle Fatigue</td>
</tr>
<tr>
<td>- Weld, High Temperature, Non-metallics</td>
</tr>
<tr>
<td><strong>Safety Factors</strong></td>
</tr>
<tr>
<td>Creep-Fatigue Interaction</td>
</tr>
<tr>
<td>Weld Fatigue</td>
</tr>
<tr>
<td><strong>Simpack</strong></td>
</tr>
<tr>
<td>- 3D Multibody Dynamics Simulation</td>
</tr>
<tr>
<td>- Mechanical or Mechatronic Systems</td>
</tr>
<tr>
<td>- Detailed Transient Simulation (Offline and Realtime)</td>
</tr>
<tr>
<td><strong>Complete System Analyses</strong></td>
</tr>
<tr>
<td>(Quasi-)Static, Dynamics, NVH</td>
</tr>
<tr>
<td>Flex Bodies, Advanced Contact</td>
</tr>
</tbody>
</table>
Join the Community!

How can you maximize the robust technology of the SIMULIA Portfolio?
Connect with peers to share knowledge and get technical insights

Go to www.3ds.com/slc to log in or join!

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.

SIMULIA SERVICES
PROVIDING HIGH QUALITY SIMULATION AND TRAINING SERVICES TO ENABLE OUR CUSTOMERS TO BE MORE PRODUCTIVE AND COMPETITIVE.

Training Schedule & Registration
We offer regularly scheduled public seminars as well as training courses at customer sites. An extensive range of courses are available, ranging from basic introductions to advanced courses that cover specific analysis topics and applications. On-site courses can be customized to focus on topics of particular interest to the customer, based on the customer's prior specification. To view the worldwide course schedule and to register for a course, visit the links below.

North American
▷ By Location
▷ By Course

International
▷ By Location
▷ By Course

Live Online Training
▷ Full Schedule
Legal Notices

The software described in this documentation is available only under license from Dassault Systèmes or its subsidiaries 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 subsidiaries.

© Dassault Systèmes, 2016

Printed in the United States of America.

Abaqus, the 3DS logo, and SIMULIA 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 SIMULIA User Assistance.
# Revision Status

<table>
<thead>
<tr>
<th>Lecture 1</th>
<th>11/16</th>
<th>Updated for Abaqus 2017</th>
</tr>
</thead>
<tbody>
<tr>
<td>Lecture 2</td>
<td>11/16</td>
<td>Updated for Abaqus 2017</td>
</tr>
<tr>
<td>Lecture 3</td>
<td>11/16</td>
<td>Updated for Abaqus 2017</td>
</tr>
<tr>
<td>Lecture 4</td>
<td>11/16</td>
<td>Updated for Abaqus 2017</td>
</tr>
<tr>
<td>Lecture 5</td>
<td>11/16</td>
<td>Updated for Abaqus 2017</td>
</tr>
<tr>
<td>Lecture 6</td>
<td>11/16</td>
<td>Updated for Abaqus 2017</td>
</tr>
<tr>
<td>Lecture 7</td>
<td>11/16</td>
<td>Updated for Abaqus 2017</td>
</tr>
<tr>
<td>Lecture 8</td>
<td>11/16</td>
<td>Updated for Abaqus 2017</td>
</tr>
<tr>
<td>Lecture 9</td>
<td>11/16</td>
<td>Updated for Abaqus 2017</td>
</tr>
<tr>
<td>Lecture 10</td>
<td>11/16</td>
<td>Updated for Abaqus 2017</td>
</tr>
<tr>
<td>Lecture 11</td>
<td>11/16</td>
<td>Updated for Abaqus 2017</td>
</tr>
<tr>
<td>Demo 1</td>
<td>11/16</td>
<td>Updated for Abaqus 2017</td>
</tr>
<tr>
<td>Demo 2</td>
<td>11/16</td>
<td>Updated for Abaqus 2017</td>
</tr>
<tr>
<td>Demo 3</td>
<td>11/16</td>
<td>Updated for Abaqus 2017</td>
</tr>
<tr>
<td>Workshop 1</td>
<td>11/16</td>
<td>Updated for Abaqus 2017</td>
</tr>
<tr>
<td>Workshop 2</td>
<td>11/16</td>
<td>Updated for Abaqus 2017</td>
</tr>
<tr>
<td>Workshop 3</td>
<td>11/16</td>
<td>Updated for Abaqus 2017</td>
</tr>
<tr>
<td>Workshop 4</td>
<td>11/16</td>
<td>Updated for Abaqus 2017</td>
</tr>
<tr>
<td>Workshop 5</td>
<td>11/16</td>
<td>Updated for Abaqus 2017</td>
</tr>
<tr>
<td>Workshop 6</td>
<td>11/16</td>
<td>Updated for Abaqus 2017</td>
</tr>
<tr>
<td>Workshop 7</td>
<td>11/16</td>
<td>Updated for Abaqus 2017</td>
</tr>
<tr>
<td>Workshop 8</td>
<td>11/16</td>
<td>Updated for Abaqus 2017</td>
</tr>
</tbody>
</table>
Lesson content:

- What is Abaqus FEA?
- Abaqus/CAE
- Abaqus/Standard and Abaqus/Explicit
Lesson 2: Introduction

Lesson content:

- Application Areas
- Statics
- Structural Processes
- Dynamics
- Lateral Buckling
- Pipe Laying
- Thermal
- Geotechnics
- Summary
- Workshop Preliminaries
- Demonstration 1: A First Look at Abaqus/CAE
- Workshop 1: Intersecting shell-pipe modeling with Abaqus/CAE

1.5 hours
Lesson content:

- Nonlinearity in Structural Mechanics
- Including Nonlinear Effects in an Abaqus Simulation
Lesson 4: Material Modeling in Abaqus

Lesson content:

- Introduction
- Metal Behavior
  - Elasticity
  - Plasticity
- Rubber Behavior
  - Assumptions
  - Hyperelastic Material Models
  - Usage Tips
- Overview of Geotechnical Materials
- Demonstration 2: Rubber Material Evaluation
Lesson 5: Structural and Solid Elements in Abaqus

Lesson content:

- Introduction
- Structural Elements in Abaqus
- Solid Elements in Abaqus
- Workshop 2: Axisymmetric Pipe Expander Example

1.5 hours
Lesson 6: Special Purpose Elements (Part 1)

Lesson content:

- Contact Elements
- Pipe-Soil Interaction Elements
- Drag Chain Elements
- Spud Can Elements
- Workshop 3: Pipeline Pull-in Analysis
- Workshop 4: Buried Pipeline Analysis

2 hours
Lesson 7: Special Purpose Elements (Part 2)

Lesson content:

- Pipe Elements
- Elbow Elements
- Axisymmetric Elements with Nonaxisymmetric Response
- Cylindrical Elements
Lesson 8: Seabed-Pipe Interaction

Lesson content:

- Seabed-Pipe Interaction
- Subroutine FRIC for Pipe-Soil Interaction
- Workshop 5: Lateral Buckling of a Sub-Sea Pipeline
- Workshop 6: Threaded Connector Analysis

1.5 hours
Lesson 9: Abaqus/Aqua

Lesson content:

- Introduction
- Defining the Offshore Environment
- Defining Offshore Loads
- Workshop 7: Abaqus/Aqua Example

1.5 hours
Lesson content:

- Coupled Eulerian-Lagrangian (CEL) Approach
- Offshore Applications of CEL
- Demonstration 3: CEL interface in Abaqus/CAE
- Workshop 8: Deformation of an Elastic Dam under Time-dependent Water Pressure
Lesson 11: Modeling Tips & Special Analysis Techniques

Lesson content:

- Random Waves
- Generating Seabed/Pipe-Wall Profiles
  - Abaqus/CAE - Lofting
- Scripting
- Modeling Suggestions: Pipe Laying
  - Contact modeling tips
- Modeling Suggestions: Surface Elements

1 hour