

Abaqus for Offshore Analysis

Abaqus 2020





3DEXPERIENCE[®]

About this Course

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 1

Lesson 1	Overview of Abaqus
Lesson 2	Introduction to Offshore Applications with Abaqus
Demo 1Workshop 1	A First Look at Abaqus/CAE Intersecting shell-pipe modeling with Abaqus/CAE
Lesson 3	What Makes a Problem Nonlinear
Lesson 4	Materials
Demo 2	Rubber Material Evaluation
Lesson 5	Structural and Solid Elements in Abaqus
Workshop 2	Axisymmetric Pipe Expander Example
Lesson 6	Special Purpose Elements (Part 1)
Workshop 3Workshop 4	Pipeline Pull-in Analysis Buried Pipeline Analysis

Day 2

- Lesson 7 Special Purpose Elements (Part 2)
- Lesson 8 Seabed Pipe Interaction
 - Workshop 5 Lateral Buckling of a Sub-Sea Pipeline
 - Workshop 6 Threaded Connector Analysis
- Lesson 9 Abaqus/Aqua Capabilities
 - Workshop 7 Riser Dynamics with Abaqus/Aqua
- Lesson 10 Coupled Eulerian-Lagrangian (CEL) Approach
 - Workshop 8 Deformation of an Elastic Dam under Time-dependent Water Pressure
- Lesson 11 Modeling Tips & Special Analysis Techniques

SIMULIA

- 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

SIMULIA's Power of the Portfolio



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 <u>www.3ds.com/slc</u> 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**. **Connect. Share. Spark Innovation.**

SIMULIA Training

http://www.3ds.com/products-services/simulia/services/training-courses/

 ▲ … ▼ SIMULIA ▼ SERVICES ▼ TRAINING COURSES ▼ 	SCHEDULE & REGISTRATION •	
3 SIMULIA	in f 🖤 🔠 🍞	
SIMULIA SERVICES PROVIDING HIGH QUALITY SIMULATION AND TRAINING SERVICES ENABLE OUR CUSTOMERS TO BE MORE PRODUCTIVE AND COMPETITIVE	S TO	

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

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, 2019

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.

Lesson 1	11/19	Updated for Abaqus 2020
Lesson 2	11/19	Updated for Abaqus 2020
Lesson 3	11/19	Updated for Abaqus 2020
Lesson 4	11/19	Updated for Abaqus 2020
Lesson 5	11/19	Updated for Abaqus 2020
Lesson 6	11/19	Updated for Abaqus 2020
Lesson 7	11/19	Updated for Abaqus 2020
Lesson 8	11/19	Updated for Abaqus 2020
Lesson 9	11/19	Updated for Abaqus 2020
Lesson 10	11/19	Updated for Abaqus 2020
Lesson 11	11/19	Updated for Abaqus 2020

Demo 1	11/19	Updated for Abaqus 2020
Demo 2	11/19	Updated for Abaqus 2020
Workshop 1	11/19	Updated for Abaqus 2020
Workshop 2	11/19	Updated for Abaqus 2020
Workshop 3	11/19	Updated for Abaqus 2020
Workshop 4	11/19	Updated for Abaqus 2020
Workshop 5	11/19	Updated for Abaqus 2020
Workshop 6	11/19	Updated for Abaqus 2020
Workshop 7	11/19	Updated for Abaqus 2020
Workshop 8	11/19	Updated for Abaqus 2020

Lesson 1: Overview of Abaqus

- ▶ What is Abaqus FEA?
- Abaqus/CAE
- Abaqus/Standard and Abaqus/Explicit

Lesson 2: Introduction

- 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



Lesson 3: Nonlinear FEA for Offshore Applications

- Nonlinearity in Structural Mechanics
- Including Nonlinear Effects in an Abaqus Simulation

Lesson 4: Material Modeling in Abaqus

- 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

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

Lesson 6: Special Purpose Elements (Part 1)

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



Lesson 7: Special Purpose Elements (Part 2)

- Pipe Elements
- Elbow Elements
- Axisymmetric Elements with Nonaxisymmetric Response
- Cylindrical Elements



Lesson 8: Seabed-Pipe Interaction

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

Lesson 9: Abaqus/Aqua

- Introduction
- Defining the Offshore Environment
- Defining Offshore Loads
- Workshop 7: Riser Dynamics with Abaqus/Aqua



Lesson 10: Coupled Eulerian-Lagrangian (CEL) Approach

- Coupled Eulerian-Lagrangian (CEL) Approach
- Offshore Applications of CEL
- Workshop 8: Deformation of an Elastic Dam under Time-dependent Water Pressure



Lesson 11: Modeling Tips & Special Analysis Techniques

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

