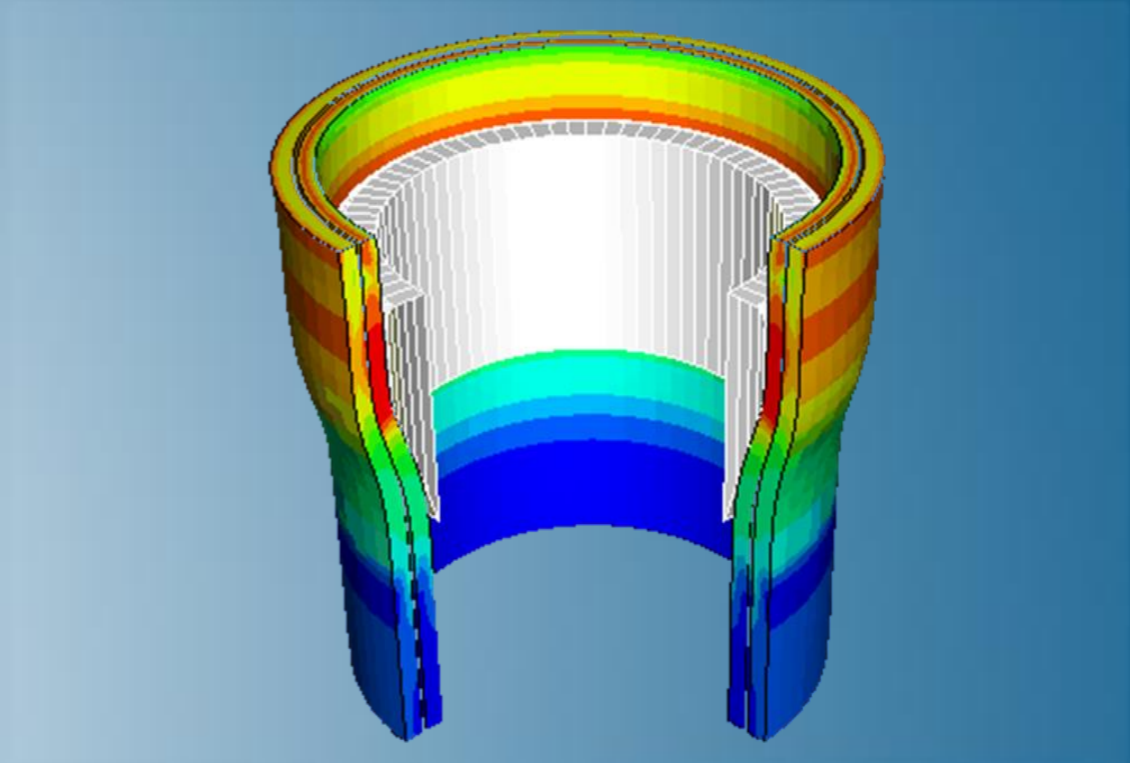


Abaqus for Offshore Analysis

Abaqus 2019



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



2 days

Day 1

- ▶ Lecture 1 Overview of Abaqus

- ▶ Lecture 2 Introduction to Offshore Applications with Abaqus
 - Demo 1 A First Look at Abaqus/CAE
 - Workshop 1 Intersecting shell-pipe modeling with Abaqus/CAE

- ▶ Lecture 3 What Makes a Problem Nonlinear

- ▶ Lecture 4 Materials
 - Demo 2 Rubber Material Evaluation

- ▶ Lecture 5 Structural and Solid Elements in Abaqus
 - Workshop 2 Axisymmetric Pipe Expander Example

- ▶ Lecture 6 Special Purpose Elements (Part 1)
 - Workshop 3 Pipeline Pull-in Analysis
 - Workshop 4 Buried Pipeline Analysis

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 Riser Dynamics with Abaqus/Aqua

- ▶ 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

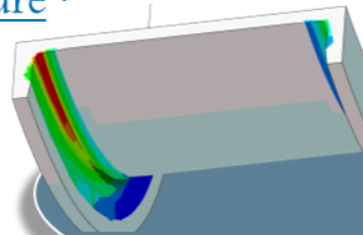
- ▶ Lecture 11 Modeling Tips & Special Analysis Techniques

SIMULIA

- ▶ SIMULIA is the Dassault Systèmes brand for Realistic Simulation solutions
- ▶ Portfolio of established, best-in-class products
 - Abaqus, Isight, Tosca, fe-safe, Simpack

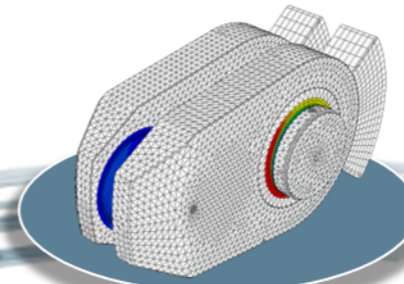
Design Optimization: Tosca Structure *

Simulation-driven design refinement to improve performance



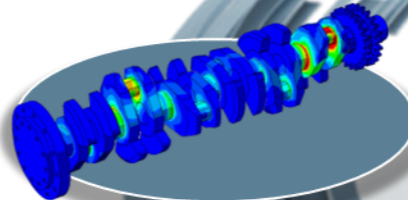
Durability Assessment: fe-safe *

Accurate life estimation to achieve certification



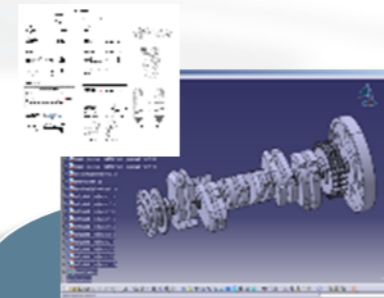
FEA Stress Analysis: Abaqus *

Detailed stress analysis using extracted load history from MBS



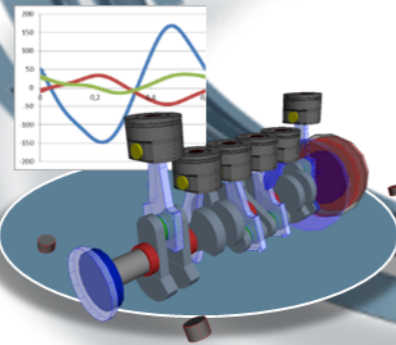
CAD Geometry: CATIA

Fully parameterized 3D geometry; FEA model generation via associative interface



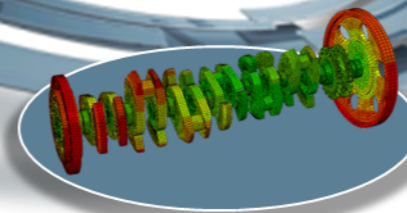
Multibody Simulation: Simpack

System analysis to extract virtual load history of complete working cycle



Mesh Calibration: Isight *

Automated mesh calibration; sufficient mesh quality for accurate results

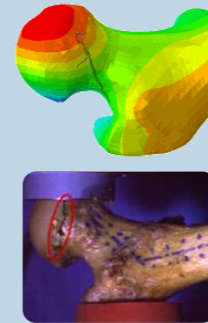


* Included in extended licensing pool

SIMULIA's Power of the Portfolio

Abaqus

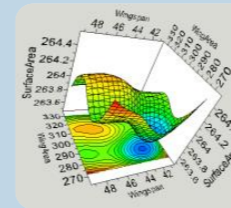
- Routine and Advanced Simulation
- Linear and Nonlinear, Static and Dynamic
- Thermal, Electrical, Acoustics
- Extended Physics through Co-simulation
- Model Preparation and Visualization



Realistic Human Simulation
High Speed Crash & Impact
Noise & Vibration

Isight

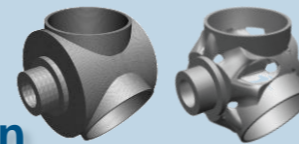
- Process Integration
- Design Optimization
- Parametric Optimization
- Six Sigma and Design of Experiments



Material Calibration
Workflow Automation
Design Exploration

Tosca

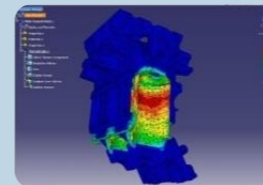
- Non-Parametric Optimization
- Structural and Fluid Flow Optimization
- Topology, Sizing, Shape, Bead Optimization



Conceptual/Detailed Design
Weight, Stiffness, Stress
Pressure Loss Reduction

fe-safe

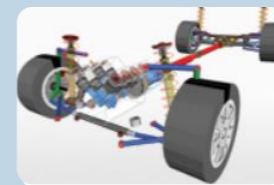
- Durability Simulation
- Low Cycle and High Cycle Fatigue
- Weld, High Temperature, Non-metallics



Safety Factors
Creep-Fatigue Interaction
Weld Fatigue

Simpack

- 3D Multibody Dynamics Simulation
- Mechanical or Mechatronic Systems
- Detailed Transient Simulation (Offline and Realtime)



Complete System Analyses
(Quasi-)Static, Dynamics, NVH
Flex Bodies, Advanced
Contact

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!



 SIMULIA

Let the SIMULIA Learning Community be *Your* Portal to 21st Century Innovation

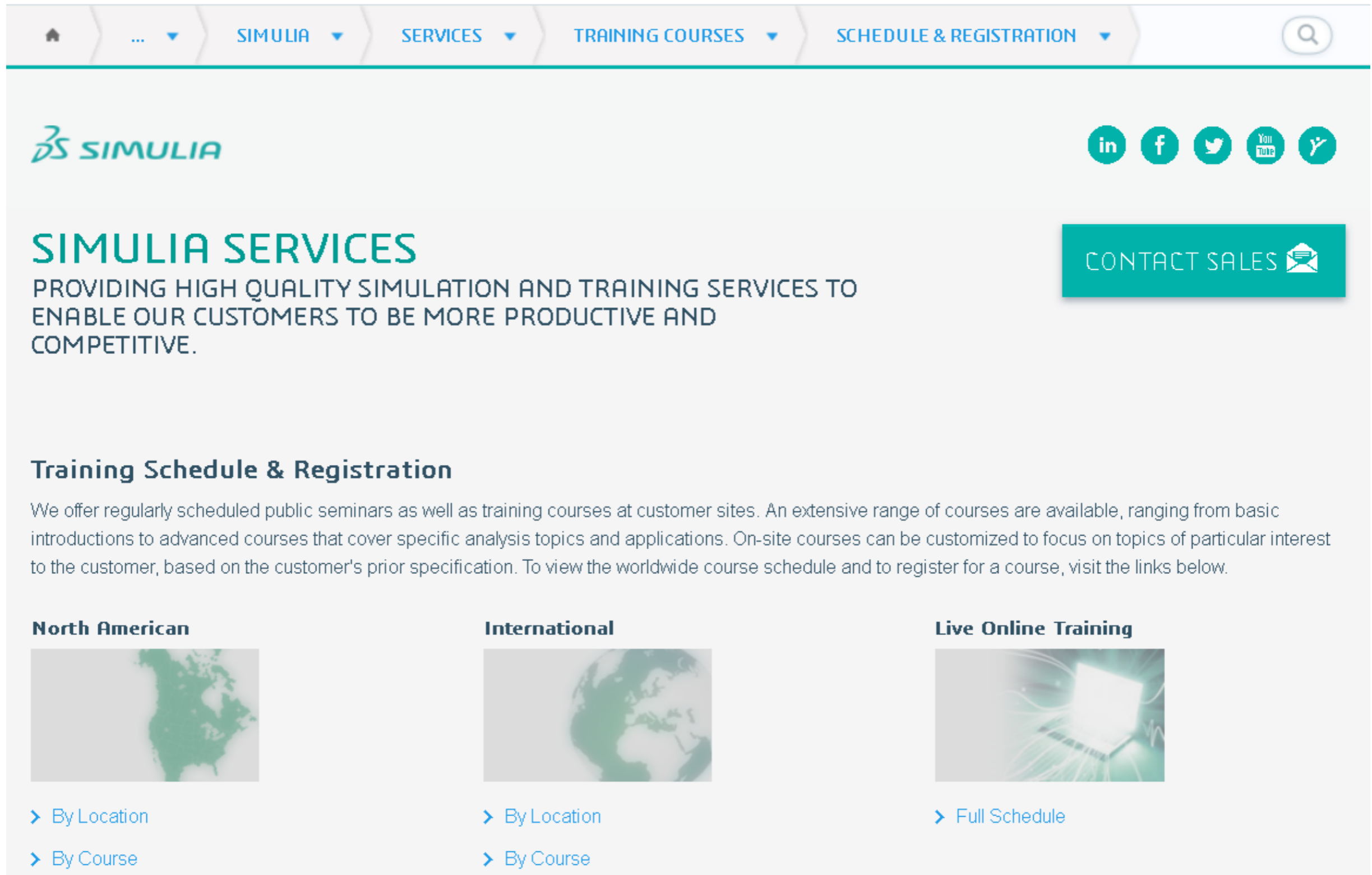
Discover new ways to explore how to leverage realistic simulation to drive product innovation. Join the thousands of Abaqus and Isight users who are already gaining valuable knowledge from the SIMULIA Learning Community.

For more information and registration, visit 3ds.com/simulia-learning.
Connect. Share. Spark Innovation.

 | The 3DEXPERIENCE Company

SIMULIA Training

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



Home ... SIMULIA SERVICES TRAINING COURSES SCHEDULE & REGISTRATION

SIMULIA

in f t YouTube

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

CONTACT SALES ✉

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, 2018

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

Lecture 1	11/18	Updated for Abaqus 2019
Lecture 2	11/18	Updated for Abaqus 2019
Lecture 3	11/18	Updated for Abaqus 2019
Lecture 4	11/18	Updated for Abaqus 2019
Lecture 5	11/18	Updated for Abaqus 2019
Lecture 6	11/18	Updated for Abaqus 2019
Lecture 7	11/18	Updated for Abaqus 2019
Lecture 8	11/18	Updated for Abaqus 2019
Lecture 9	11/18	Updated for Abaqus 2019
Lecture 10	11/18	Updated for Abaqus 2019
Lecture 11	11/18	Updated for Abaqus 2019

Demo 1	11/18	Updated for Abaqus 2019
Demo 2	11/18	Updated for Abaqus 2019
Demo 3	11/18	Updated for Abaqus 2019
Workshop 1	11/18	Updated for Abaqus 2019
Workshop 2	11/18	Updated for Abaqus 2019
Workshop 3	11/18	Updated for Abaqus 2019
Workshop 4	11/18	Updated for Abaqus 2019
Workshop 5	11/18	Updated for Abaqus 2019
Workshop 6	11/18	Updated for Abaqus 2019
Workshop 7	11/18	Updated for Abaqus 2019
Workshop 8	11/18	Updated for Abaqus 2019

Lesson 1: Overview of Abaqus

Lesson content:

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



15 minutes

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 3: Nonlinear FEA for Offshore Applications

Lesson content:

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



15 minutes

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



1 hour

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



30 minutes

Lesson 8: Seabed-Pipe Interaction

Lesson content:

- ▶ 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: Riser Dynamics with Abaqus/Aqua



1.5 hours

Lesson 10: Coupled Eulerian-Lagrangian (CEL) Approach

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



1.5 hours

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