

Learning Module

Abaqus for Offshore Analysis

This course was designed by SIMULIA UK in support of their offshore customers to provide them a more indepth, industry-specific training. The workshops in this course are completely new and were developed from customer applications.

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.

Knowledge Prerequisites

None

Brands

Simulia

Available Releases

SIMULIA 2021, SIMULIA 2020, SIMULIA 2019, SIMULIA 2018, SIMULIA 2017, SIMULIA 2016, SIMULIA V6.14, SIMULIA V6.13, SIMULIA V6.12

Duration

16 hours

Discipline

Advanced Abaqus

Language(s) for selected release

English

Contents

Overview - Abaqus for Offshore Analysis

1 - Overview of Abaqus

2 - Introduction

3 - Nonlinear FEA for Offshore Applications

4 - Material Modeling in Abaqus

5 - Structural and Solid Elements in Abaqus

6 - Special Purpose Elements (Part 1)

7 - Special Purpose Elements (Part 2)

8 - Seabed-Pipe Interaction

9 - Abaqus Aqua

10 - Coupled Eulerian-Lagrangian (CEL) Approach

11 - Modeling Tips & Special Analysis Techniques