Fitness-for-Service Analysis with Abaqus

Abaqus 2020
Course objectives

Upon completion of this course you will be able to:

- Use Abaqus/CAE to create finite element models of common plant structures.
- Use Abaqus/CAE to submit and monitor analysis jobs.
- Use Abaqus to perform buckling, elastic-plastic analyses.

Targeted audience

Fixed–Equipment Reliability Engineers

Prerequisites

None

About this Course

2 days
<table>
<thead>
<tr>
<th>Day 1</th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Lesson 1</td>
<td>Overview of Abaqus</td>
</tr>
<tr>
<td>Lesson 2</td>
<td>Introduction to Abaqus/CAE – Part I</td>
</tr>
<tr>
<td><strong>Workshop 1</strong></td>
<td><strong>Modeling of a vertical tower</strong></td>
</tr>
<tr>
<td>Lesson 3</td>
<td>Introduction to Abaqus/CAE – Part II</td>
</tr>
<tr>
<td><strong>Workshop 2</strong></td>
<td><strong>Modeling of a flange joint</strong></td>
</tr>
<tr>
<td>Lesson 4</td>
<td>Introduction to Abaqus/CAE – Part III</td>
</tr>
<tr>
<td><strong>Workshop 3</strong></td>
<td><strong>Analysis of a vertical tower with rings</strong></td>
</tr>
<tr>
<td>Lesson 5</td>
<td>Overview of Abaqus solvers</td>
</tr>
<tr>
<td><strong>Workshop 4</strong></td>
<td><strong>Analysis of flange connection</strong></td>
</tr>
</tbody>
</table>
Day 2

Lesson 6  Overview of Fitness-for-Service analyses
Lesson 7  Applying loads in a Simulation
  Workshop 5  Modeling and analysis of spherical storage vessel
Lesson 8  Analysis of metal loss
  Workshop 6  Modeling and analysis of a horizontal vessel
Lesson 9  Analysis of a flange connection
  Workshop 7  Limit load of a heat exchanger
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

- FEA Stress Analysis: Abaqus *
  - Detailed stress analysis using extracted load history from MBS

- Multibody Simulation: Simpack
  - System analysis to extract virtual load history of complete working cycle

- Durability Assessment: fe-safe *
  - Accurate life estimation to achieve certification

- 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

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

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

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

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

**Realistic Human Simulation**
- High Speed Crash & Impact
- Noise & Vibration

**Material Calibration**
- Workflow Automation
- Design Exploration

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

**Safety Factors**
- Creep-Fatigue Interaction
- Weld Fatigue

**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!

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.

©2013 Dassault Systèmes. All rights reserved.
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, 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.
<table>
<thead>
<tr>
<th>Lesson/Workshop</th>
<th>Date</th>
<th>Update Details</th>
</tr>
</thead>
<tbody>
<tr>
<td>Lesson 1</td>
<td>11/19</td>
<td>Updated for Abaqus 2020</td>
</tr>
<tr>
<td>Lesson 2</td>
<td>11/19</td>
<td>Updated for Abaqus 2020</td>
</tr>
<tr>
<td>Lesson 3</td>
<td>11/19</td>
<td>Updated for Abaqus 2020</td>
</tr>
<tr>
<td>Lesson 4</td>
<td>11/19</td>
<td>Updated for Abaqus 2020</td>
</tr>
<tr>
<td>Lesson 5</td>
<td>11/19</td>
<td>Updated for Abaqus 2020</td>
</tr>
<tr>
<td>Lesson 6</td>
<td>11/19</td>
<td>Updated for Abaqus 2020</td>
</tr>
<tr>
<td>Lesson 7</td>
<td>11/19</td>
<td>Updated for Abaqus 2020</td>
</tr>
<tr>
<td>Lesson 8</td>
<td>11/19</td>
<td>Updated for Abaqus 2020</td>
</tr>
<tr>
<td>Lesson 9</td>
<td>11/19</td>
<td>Updated for Abaqus 2020</td>
</tr>
<tr>
<td>Workshop 1</td>
<td>11/19</td>
<td>Updated for Abaqus 2020</td>
</tr>
<tr>
<td>Workshop 2</td>
<td>11/19</td>
<td>Updated for Abaqus 2020</td>
</tr>
<tr>
<td>Workshop 3</td>
<td>11/19</td>
<td>Updated for Abaqus 2020</td>
</tr>
<tr>
<td>Workshop 4</td>
<td>11/19</td>
<td>Updated for Abaqus 2020</td>
</tr>
<tr>
<td>Workshop 5</td>
<td>11/19</td>
<td>Updated for Abaqus 2020</td>
</tr>
<tr>
<td>Workshop 6</td>
<td>11/19</td>
<td>Updated for Abaqus 2020</td>
</tr>
<tr>
<td>Workshop 7</td>
<td>11/19</td>
<td>Updated for Abaqus 2020</td>
</tr>
</tbody>
</table>
Lesson content:

- What is Abaqus FEA?
- Abaqus/CAE
- Abaqus Solvers
- Abaqus Conventions
- Working with the Model Tree
- Other Abaqus/CAE Topics
- Documentation
- SIMULIA Learning Community

30 minutes
Lesson content:

- Abaqus/CAE: Part Module
  - What are Parts?
  - Defining a Part
  - Geometry Import and Repair
  - Building a Part Using the Part Module Tools
- Abaqus/CAE: Property Module
  - Overview of Abaqus Material Models
  - Linear Elasticity
  - Metal Plasticity
  - Section Properties
  - Material Databases
- Abaqus/CAE: Assembly Module
  - What is an Assembly?
  - Positioning Instances
- Workshop Preliminaries
- Workshop 1: Modeling of a Vertical Tower
1. In this workshop, instructions on how to model a vertical tower in Abaqus/CAE are provided.

2. The tower will be used to perform the following analyses:
   a. Buckling load prediction with preload
   b. Buckling load prediction without preload

Workshop 1: Modeling of a Vertical Tower

45 minutes
Lesson 3: Introduction to Abaqus/CAE – Part II

Lesson content:

- Abaqus/CAE: Step Module
  - Analysis Steps and Procedures
  - Output
- Abaqus/CAE: Interaction Module
  - Tie Constraints
- Abaqus/CAE: Load Module
  - Loads and Boundary Conditions
  - Model Verification
- Abaqus/CAE: Mesh Module
  - What is a Mesh?
  - Elements in Abaqus
  - Mesh Generation Workflow
  - The Mesh Module
  - Common Tools:
    - Density
    - Controls
    - Element Selection
    - Meshing
    - Local Fine-tuning
    - Quality Checks
- Workshop 2: Modeling of a Flange Connection

1 hour
1. In this workshop, instructions on how to model a flange connection in Abaqus/CAE are provided.

2. Several modeling techniques to create a solid element model of the flange will be discussed.
   
a. In a later exercise, the flange will be used to perform bolt-up analysis.
Lesson content:

- Abaqus/CAE: Job Module
  - Job Management
- Abaqus/CAE: Visualization Module
  - Results Visualization: Basic features
  - Results Visualization: Advanced/miscellaneous features
- Workshop 3: Analysis of a Vertical Tower with Rings
1. This workshop provides instructions on how to add vacuum rings to a vertical tower using Abaqus/CAE

2. This workshop is intended to augment the previously generated model of the vertical tower

3. The tower will be used to perform the following analyses
   a. Buckling load prediction with preload
   b. Buckling load prediction without preload
   c. Buckling load prediction with preloading and NLGEOM
Lesson 5: Overview of Solvers

Lesson content:

- Introduction
- Abaqus Usage
- Example: Column Buckling
- Nonlinearity in Structural Mechanics
- Nonlinear Analysis using Implicit Methods
- Including Nonlinear Effects in an Abaqus Simulation
- Convergence Issues
- Workshop 4: Analysis of a Flange Connection
1. This workshop provides instructions on analyzing a flange connection in Abaqus/CAE

2. This workshop is intended to start from a previously generated model of the flange

3. The flange will be used to perform the following analyses
   
   a. Response prediction after applying bolt loading
   
   b. Response prediction after applying bolt load and internal pressure
Lesson content:

- Overview of Fitness for Service assessment
- ASME/API 579 – Overview
- Collapse analysis of a corroded pipe
- Analysis of a pipe with an inner flaw
- Analysis of a pipe with a gouge
Lesson 7: Applying Loads in a Simulation

Lesson content:

- Common types of loads in a Fitness-for-Service assessment
- Applying and verifying loads
- Applying dead weight
- Applying pressure
- Applying static head
- Applying wind load
- Applying thermal load
- Applying multiple loads simultaneously
- Workshop 5: Modeling and Analysis of a Spherical Vessel

1.5 hours
1. This workshop provides instructions on how to model a spherical vessel in Abaqus/CAE

2. The vessel will be used to perform the following analyses
   
a. Static stress with weight of the contents and internal pressure
Lesson content:

- Analysis methods for protection against plastic collapse
- Loads in a Fitness-for-Service assessment
- Limit load analysis
  - Characteristics
  - Procedure
- Limit load analysis of a vertical vessel
- Elastic-plastic analysis
  - Characteristics
  - Procedure
- Elastic-plastic analysis of a vertical vessel
- Thickness mapping tool using Abaqus/CAE
- Workshop 6: Modeling and Analysis of a Horizontal Vessel
1. This workshop provides instructions on how to model a horizontal vessel in Abaqus/CAE

2. The vessel will be used to perform the following analyses
   
a. Static stress with weight of the internal fluids
Lesson 9: Analysis of a Flange Connection

Lesson content:

- Abaqus/CAE: Interaction Module
- Basic Architecture of a Flange Connection
- Analysis Requirements
- What is Contact?
- Approaches to Modeling Contact
- Defining General Contact
- Defining Contact Pairs
- Features of a Contact Formulation
- Output of Contact Results
- Modeling Gasket Behavior
- Analysis of a Bolted Flange Joint
- Workshop 7: Limit Load of a Heat Exchanger

1 hour
1. Instructions are provided for creating a heat exchanger model and mapping thickness loss using Abaqus/CAE

2. The heat exchanger will be used to perform the following analyses
   a. Limit load prediction with uniform corrosion
   b. Limit load prediction with thickness data

3. Abaqus/CAE plug-in: The Thickness Mapping Tool needs to be installed to complete this workshop (plug-in installation files, installation instructions document and an usage guide with an illustrative example can be obtained from the SIMULIA Learning Community)