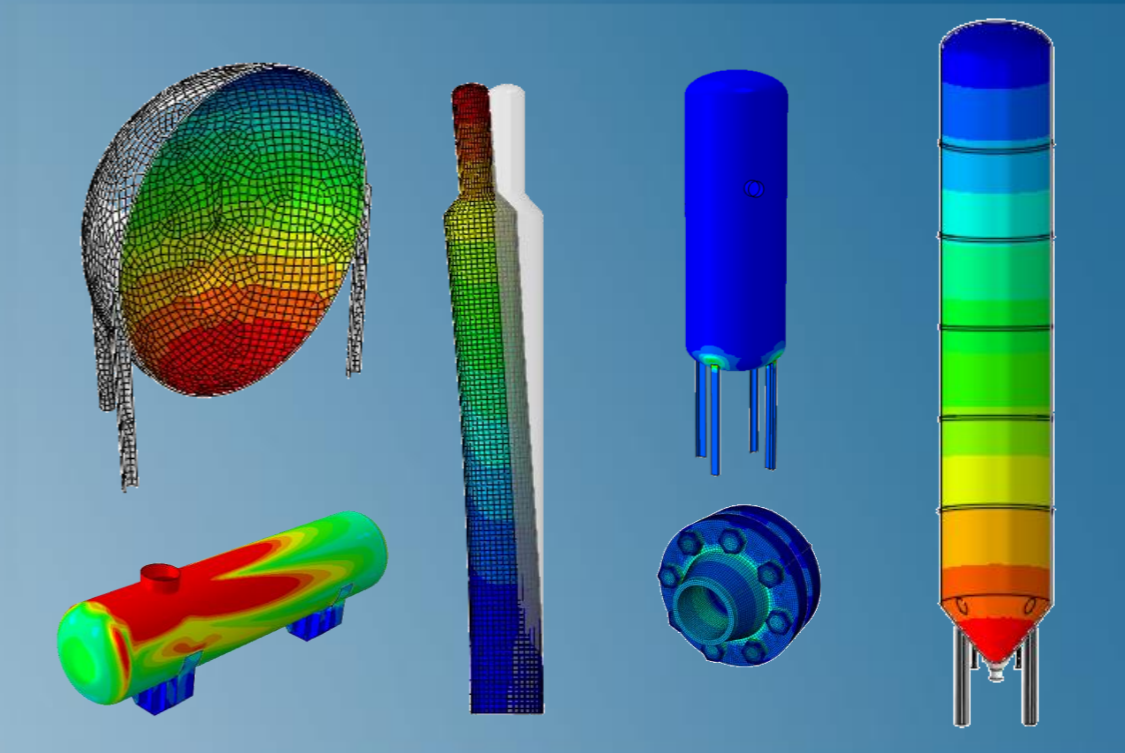


# Fitness-for-Service Analysis with Abaqus

Abaqus 2022



**3DEXPERIENCE®**

# About this Course

## 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



2 days

# Day 1

---

Lesson 1

Overview of Abaqus

Lesson 2

Introduction to Abaqus/CAE – Part I

## **Workshop 1**

**Modeling of a vertical tower**

Lesson 3

Introduction to Abaqus/CAE – Part II

## **Workshop 2**

**Modeling of a flange joint**

Lesson 4

Introduction to Abaqus/CAE – Part III

## **Workshop 3**

**Analysis of a vertical tower with rings**

Lesson 5

Overview of Abaqus solvers

## **Workshop 4**

**Analysis of flange connection**

## 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

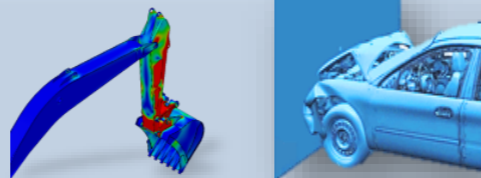
### **Workshop 7**

**Limit load of a heat exchanger**

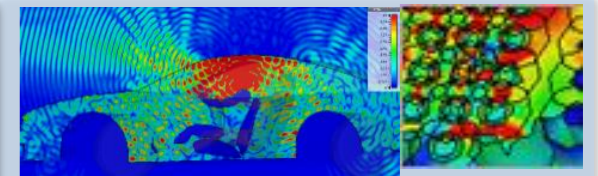
# SIMULIA

- ▶ SIMULIA is the Dassault Systèmes brand for realistic simulation solutions.
- ▶ Advanced simulation portfolio covering simulation disciplines such as structural mechanics, computational fluid dynamics and electromagnetic field simulation, for a true multiphysics simulation approach.

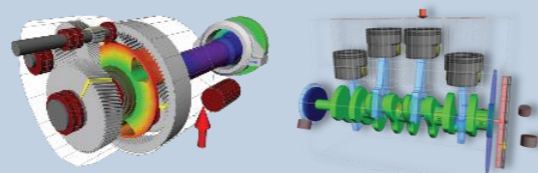
## Structures



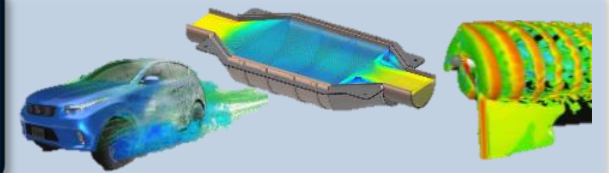
## Electromagnetics



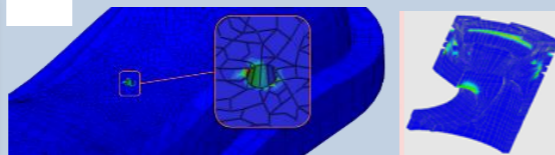
## Multibody



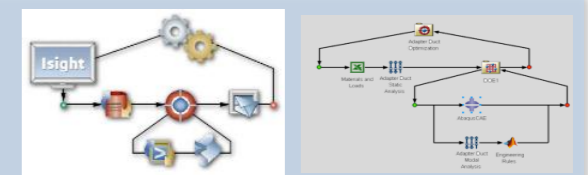
## Fluids



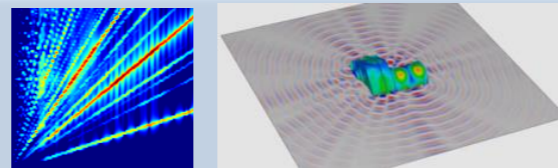
## Durability



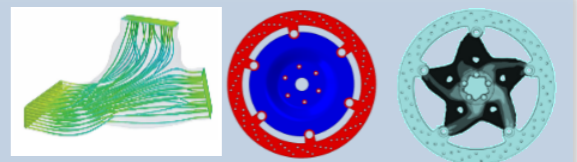
## Automation



## Vibro-acoustics



## Optimization



# Join the Community!

Go to [go.3ds.com/simc](http://go.3ds.com/simc) to log in or join!



## SIMULIA COMMUNITY

BECOME PART OF A GLOBAL USER COMMUNITY FOCUSED ON ADVANCING THE USE OF SIMULIA SIMULATION SOLUTIONS IN SCIENCE AND ENGINEERING

[LOGIN NOW](#)

### Join Us

Interested in the latest in simulation? Looking for advice and best practices? Want to discuss simulation with fellow users and Dassault Systèmes experts?

The SIMULIA Community is the place to be.

Simply [log in](#) with your 3DS Passport username and password. If you use DSx Client Care for technical support, you can use these same credentials to access the community.

If you do not already have a 3DS Passport, you can [register now](#). An account is free and access is instant.



### Join the conversation

Start a discussion with other members of the SIMULIA Community. Talk through your burning simulation questions with peers, SIMULIA experts and SIMULIA Champions. Apply to be an author to create posts, share useful tips you've discovered for SIMULIA software and establish yourself as a thought-leader. The SIMULIA Community is home to both SIMULIA product users across the world, and to SIMULIA subject matter experts.

### Stay up to date on the latest news

Modern industry trends change rapidly, and SIMULIA is always developing its products to stay ahead. Follow the SIMULIA Community to be informed of new product releases and updates to the Knowledge Base, and to receive links to articles and blog posts about the latest industry trends.



### Browse e-learning resources

The SIMULIA Community brings together learning materials covering numerous applications for SIMULIA products. Read a whitepaper on the benefits of simulation in your work, discover tips and tricks for using SIMULIA software efficiently, or watch a demonstration of how to use simulation to achieve your goals.

# SIMULIA Training

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



## SIMULIA TRAINING

PROVIDING TRAINING SERVICES TO ENABLE OUR CUSTOMERS TO BE MORE PRODUCTIVE AND COMPETITIVE

FIND A BUSINESS PARTNER

### Simulation Training

SIMULIA and our education partners 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. The same courseware, and other content, is available for self-paced eLearning. 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, register for a course, or to learn more about our eLearning options, visit the links below.

### SIMULIA DIRECT TRAINING



Instructor-lead training of both off-the-shelf materials and customized content based on your needs.

### MENTORING



Mentoring consists of short-term engagements to accelerate the efficiency and effectiveness of your processes

### EDUCATION PARTNER TRAINING



SIMULIA has a large eco-system of education partners with certified instructors who also

### SIMULIA ELEARNING RESOURCES



SIMULIA provides extensive eLearning solutions, published on various platforms, to enable:

## 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, 2021

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

---

<b>Lesson 1</b>	<b>11/21</b>	<b>Updated for Abaqus 2022</b>
<b>Lesson 2</b>	<b>11/21</b>	<b>Updated for Abaqus 2022</b>
<b>Lesson 3</b>	<b>11/21</b>	<b>Updated for Abaqus 2022</b>
<b>Lesson 4</b>	<b>11/21</b>	<b>Updated for Abaqus 2022</b>
<b>Lesson 5</b>	<b>11/21</b>	<b>Updated for Abaqus 2022</b>
<b>Lesson 6</b>	<b>11/21</b>	<b>Updated for Abaqus 2022</b>
<b>Lesson 7</b>	<b>11/21</b>	<b>Updated for Abaqus 2022</b>
<b>Lesson 8</b>	<b>11/21</b>	<b>Updated for Abaqus 2022</b>
<b>Lesson 9</b>	<b>11/21</b>	<b>Updated for Abaqus 2022</b>
<b>Workshop 1</b>	<b>11/21</b>	<b>Updated for Abaqus 2022</b>
<b>Workshop 2</b>	<b>11/21</b>	<b>Updated for Abaqus 2022</b>
<b>Workshop 3</b>	<b>11/21</b>	<b>Updated for Abaqus 2022</b>
<b>Workshop 4</b>	<b>11/21</b>	<b>Updated for Abaqus 2022</b>
<b>Workshop 5</b>	<b>11/21</b>	<b>Updated for Abaqus 2022</b>
<b>Workshop 6</b>	<b>11/21</b>	<b>Updated for Abaqus 2022</b>
<b>Workshop 7</b>	<b>11/21</b>	<b>Updated for Abaqus 2022</b>

# Lesson 1: Overview of Abaqus

## *Lesson content:*

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



30 minutes

# Lesson 2: Introduction to Abaqus/CAE – Part I

## *Lesson content:*

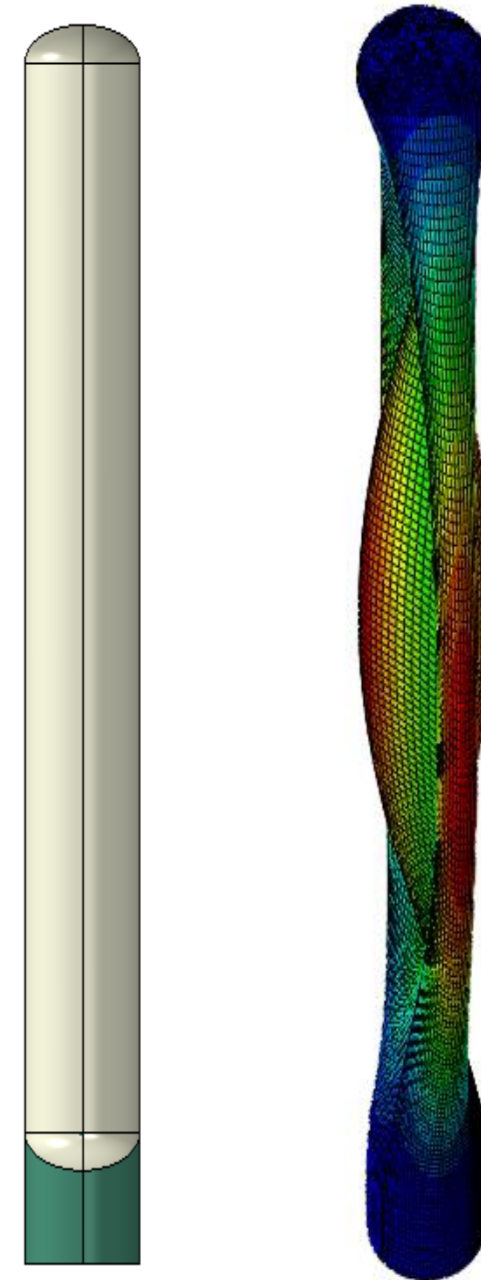
- ▶ Abaqus/CAE: Part Module
  - ❑ What are Parts?
  - ❑ Defining a Part
  - ❑ Geometry Import and Repair
  - ❑ Building a Part Using the Part Module Tools
- ▶ Abaqus/CAE: Property Module
  - ❑ Overview of Abaqus Material Models
  - ❑ Linear Elasticity
  - ❑ Metal Plasticity
  - ❑ Section Properties
  - ❑ Material Databases
- ▶ Abaqus/CAE: Assembly Module
  - ❑ What is an Assembly?
  - ❑ Positioning Instances
- ▶ Workshop Preliminaries
- ▶ Workshop 1: Modeling of a Vertical Tower



45 minutes

# 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



45 minutes

# Lesson 3: Introduction to Abaqus/CAE – Part II

## *Lesson content:*

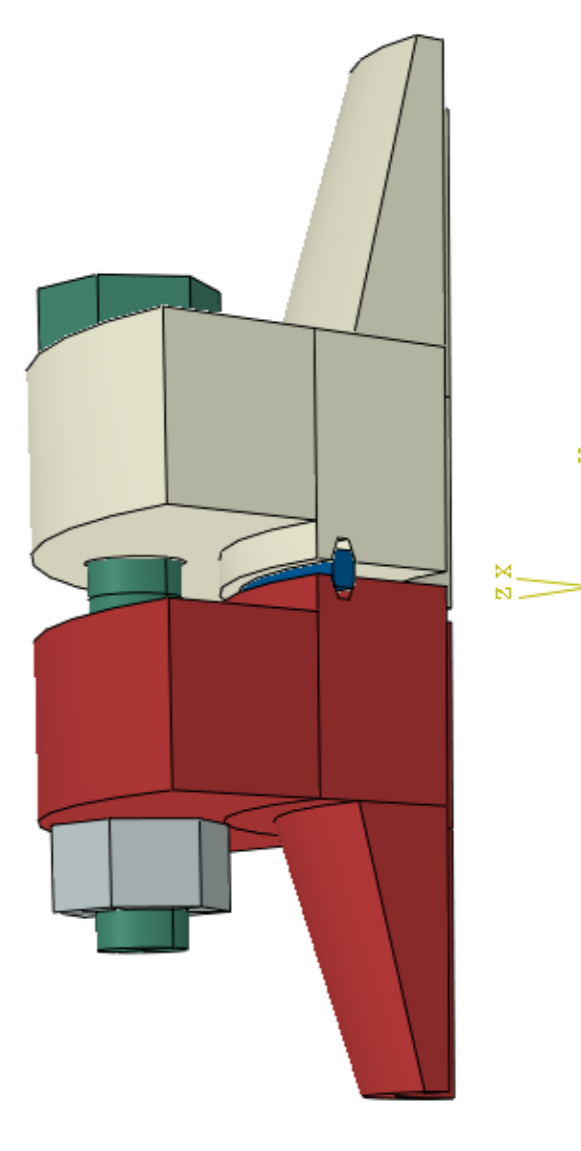
- ▶ Abaqus/CAE: Step Module
  - Analysis Steps and Procedures
  - Output
- ▶ Abaqus/CAE: Interaction Module
  - Tie Constraints
- ▶ Abaqus/CAE: Load Module
  - Loads and Boundary Conditions
  - Model Verification
- ▶ Abaqus/CAE: Mesh Module
  - What is a Mesh?
  - Elements in Abaqus
  - Mesh Generation 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

# Workshop 2: Modeling of a Flange Connection

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



1 hour

# Lesson 4: Introduction to Abaqus/CAE – Part III

## *Lesson content:*

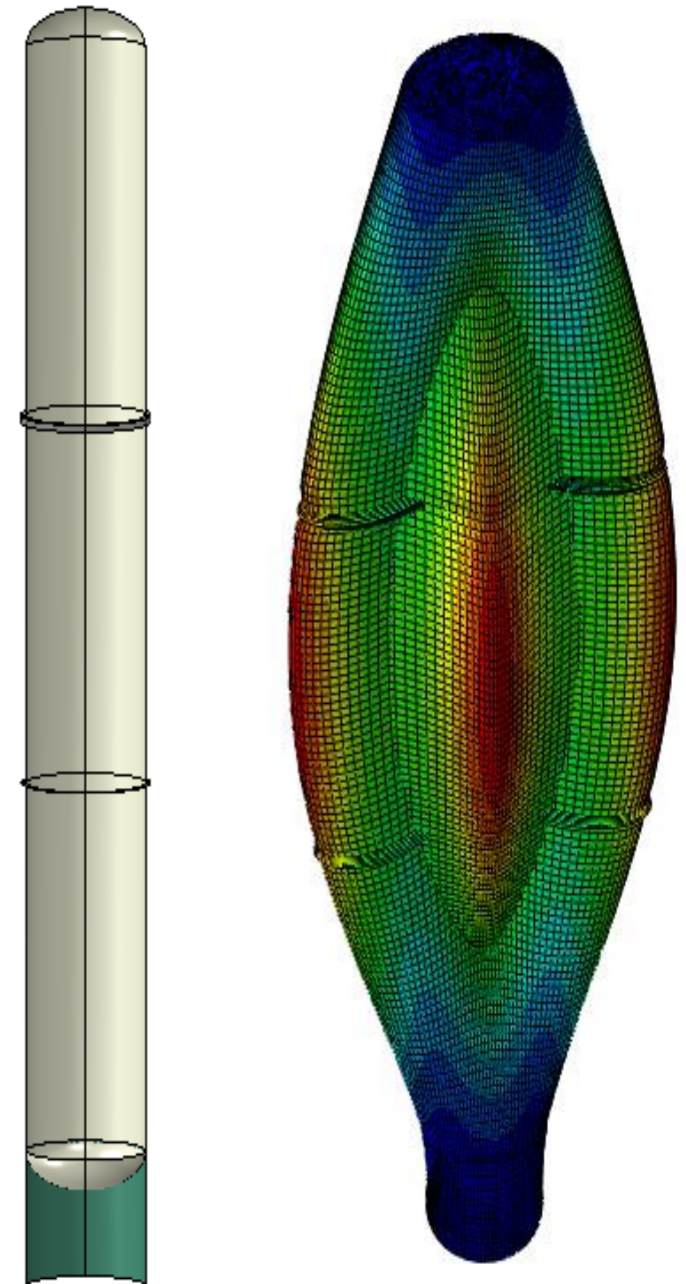
- ▶ Abaqus/CAE: Job Module
  - Job Management
- ▶ Abaqus/CAE: Visualization Module
  - Results Visualization: Basic features
  - Results Visualization: Advanced/miscellaneous features
- ▶ Workshop 3: Analysis of a Vertical Tower with Rings



45 minutes

# Workshop 3: Modeling 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



45 minutes

# Lesson 5: Overview of Solvers

## *Lesson content:*

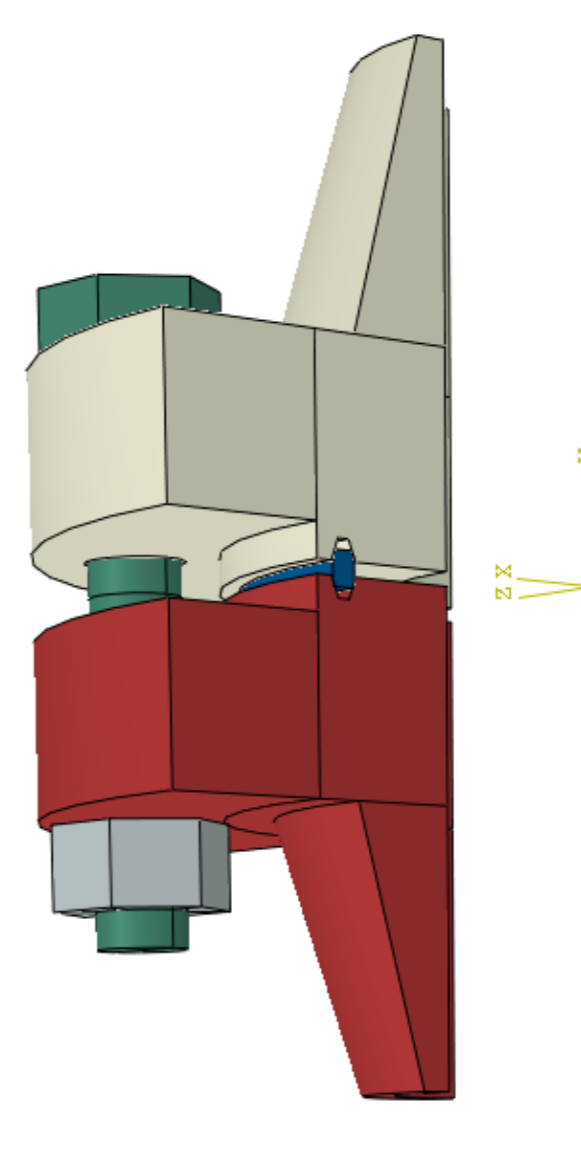
- ▶ Eigenvalue Buckling Analysis
- ▶ Abaqus Usage
- ▶ Example: Column Buckling
- ▶ Nonlinearity in Structural Mechanics
- ▶ Nonlinear Analysis using Implicit Methods
- ▶ Including Nonlinear Effects in an Abaqus Simulation
- ▶ Convergence Issues
- ▶ Workshop 4: Analysis of a Flange Connection



1 hour

# 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 6: Overview of Fitness-for-Service Analyses

## *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



30 minutes

# Lesson 7: Applying Loads in a Simulation

## *Lesson content:*

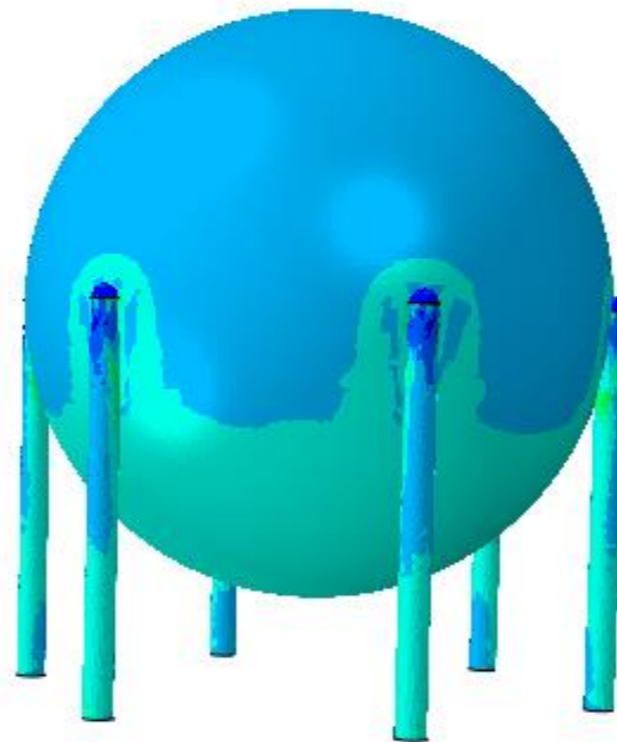
- ▶ Common types of loads in a Fitness-for-Service assessment
- ▶ Applying and verifying loads
- ▶ Applying dead weight
- ▶ Applying pressure
- ▶ Applying static head
- ▶ Applying wind load
- ▶ Applying thermal load
- ▶ Applying multiple loads simultaneously
- ▶ Workshop 5: Modeling and Analysis of a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



1.5 hours

# Workshop 5: Modeling and Analysis of a Spherical Vessel

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



60 minutes

# Lesson 8: Analysis of Metal Loss

## *Lesson content:*

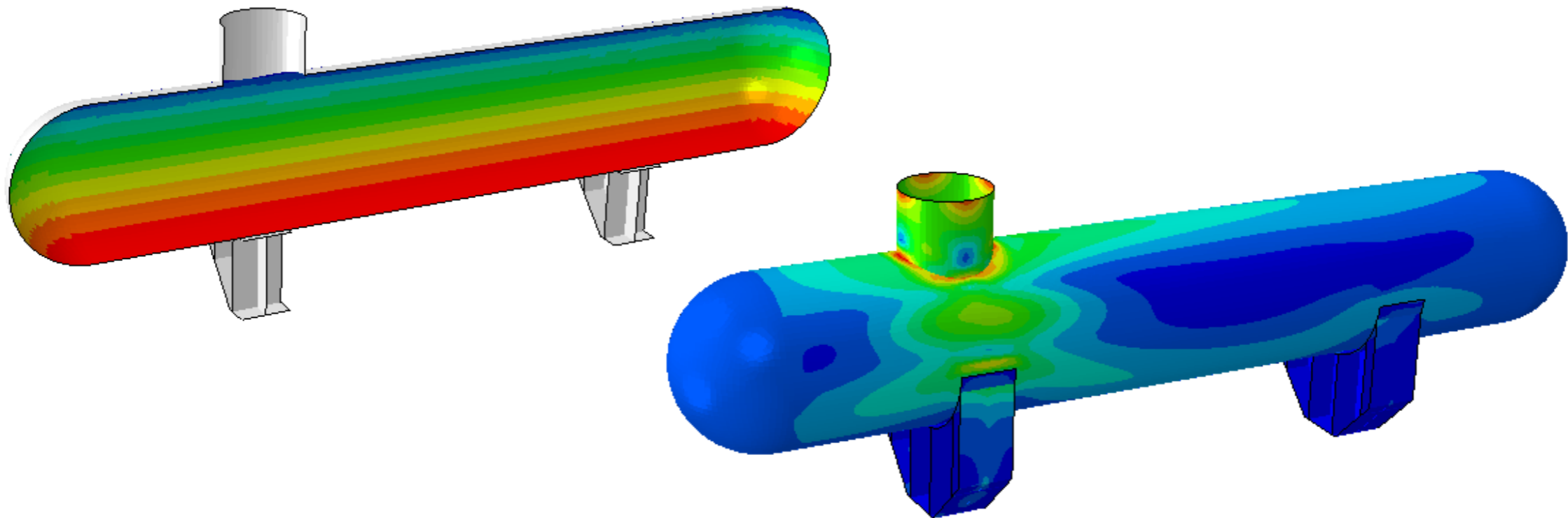
- ▶ Analysis methods for protection against plastic collapse
- ▶ Loads in a Fitness-for-Service assessment
- ▶ Limit load analysis
  - Characteristics
  - Procedure
- ▶ Limit load analysis of a vertical vessel
- ▶ Elastic-plastic analysis
  - Characteristics
  - Procedure
- ▶ Elastic-plastic analysis of a vertical vessel
- ▶ Thickness mapping tool using Abaqus/CAE
- ▶ Workshop 6: Modeling and Analysis of a Horizontal Vessel



1 hour

# Workshop 6: Modeling and Analysis of a Horizontal Vessel

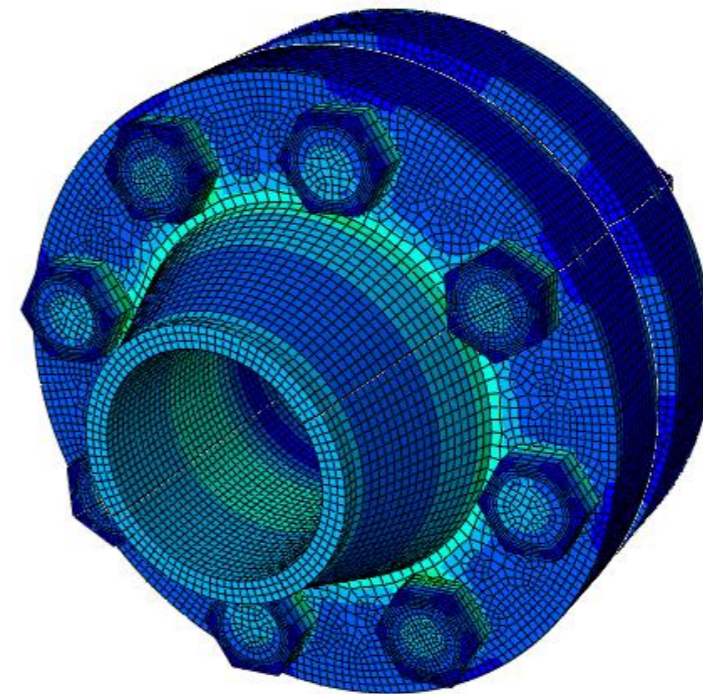
1. This workshop provides instructions on how to model a horizontal vessel in Abaqus/CAE
2. The vessel will be used to perform the following analyses
  - a. Static stress with weight of the internal fluids



# 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

# Workshop 7: Limit Load of a Heat Exchanger

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 Community)

