S VERTICAL
SIMULIA Training Catalogue 2020

Version: 1.0
Edition: 01 01 2019

Table of content

1. S-VERTICAL training offer overview:

This quotation provides the price for a private training given by S VERTICAL in the customer offices.

The final training cost will be equal to: Base Cost + (Number Of Attendees X Cost Per Attendee)

This quotation doesn’t include traveling and accommodation cost of the trainer.

Academic customers have 30% discount.

SIMULIA Training	Duration (Days)
Introduction to Abaqus	4
Modeling Contact with Abaqus/Standard	2
Obtaining a Converged Solution with Abaqus	2
Modeling Contact and Resolving Convergence with Abaqus	3
Abaqus/Explicit Advanced Topics	3
Analysis of Composite Materials with Abaqus	3
Abaqus for Offshore Analysis	2
Modeling of Geotechnical Problems with Abaqus	2
Writing User Subroutine with Abaqus	2
Python scripting in Abaqus	2
Advanced Abaqus Scripting	2
Introduction to Isight	2
Introduction to fe-safe	2
Non-Parametric optimization using Abaqus and Tosca	2

2. SIMULIA trainings description:

2.1. Introduction to Abaqus

Course objectives:

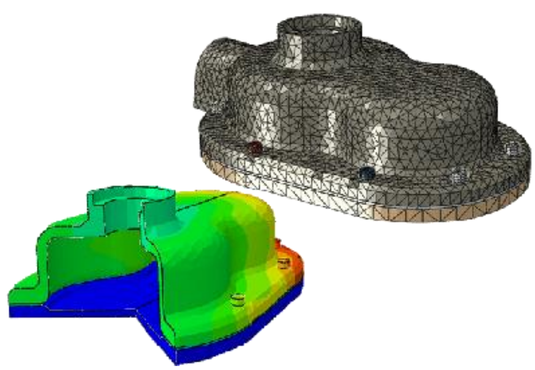
Upon completion of this course you will be able to:

- Use Abaqus/CAE to create complete finite element models.
- Use Abaqus/CAE to submit and monitor analysis jobs.
- Use Abaqus/CAE to view and evaluate simulation results.
- Solve structural analysis problems using Abaqus/Standard and Abaqus/Explicit, including the effects of material nonlinearity, large deformation and contact.

Targeted audience: Simulation Analysts

Prerequisites: None

Training duration: **4 days**



1.1. Modeling Contact with Abaqus/Standard

This introductory course is the ideal way to obtain a working knowledge of how to use both Abaqus/Standard and Abaqus/Explicit to solve linear and nonlinear problems. The seminar introduces you to the analysis capabilities of Abaqus using the keywords interface.

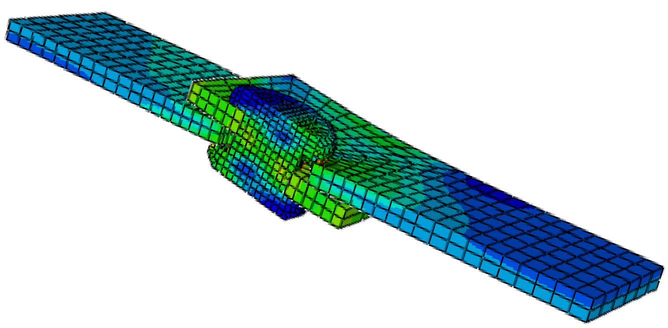
Course objectives:
Upon completion of this course you will be able to:

- Complete finite element models using Abaqus keywords.
- Submit and monitor analysis jobs.
- View and evaluate simulation results.
- Solve structural analysis problems using Abaqus/Standard and Abaqus/Explicit, including the effects of material nonlinearity, large deformation and contact.

Targeted audience: Simulation Analysts

Prerequisites: This course is recommended for engineers with experience using Abaqus/Standard

Training duration: 2 days



1.2. Obtaining a Converged Solution with Abaqus

Course objectives:

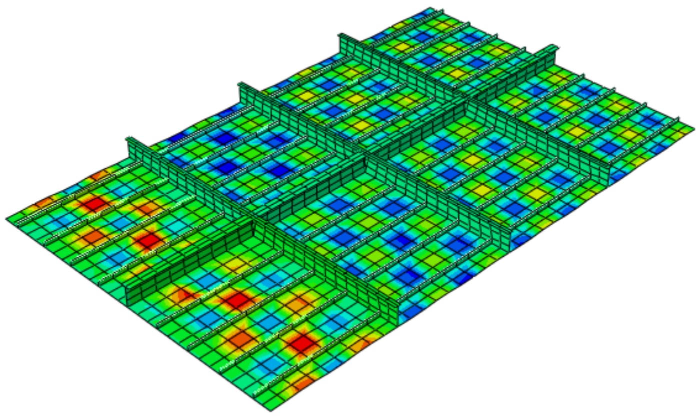
Upon completion of this course you will be able to:

- Understand how nonlinear problems are solved in Abaqus
- Develop Abaqus models that will converge
- Identify modeling errors that cause models to experience convergence difficulties
- Recognize when a problem is too difficult or too ill-posed to be solved effectively

Targeted audience: Simulation Analysts

Prerequisites: This course is recommended for engineers with experience using Abaqus

Training duration: 2 days



1.3. Modeling Contact and Resolving Convergence with Abaqus

Course objectives:

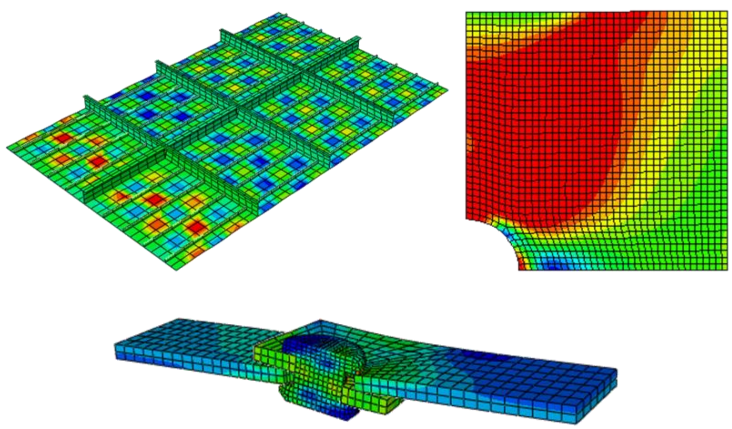
Upon completion of this course you will be able to:

- Define general contact and contact pairs
- Define appropriate surfaces (rigid or deformable)
- Model frictional contact
- Model large sliding between deformable bodies
- Resolve overclosures in interference fit problems
- Understand how nonlinear problems are solved in Abaqus
- Develop Abaqus models that will converge
- Identify modeling errors that cause models to experience convergence difficulties
- Recognize when a problem is too difficult or too ill-posed to be solved effectively

Targeted audience: Simulation Analysts

Prerequisites: This course is recommended for engineers with experience using Abaqus

Training duration: **3 days**



1.4. Abaqus/Explicit: Advanced topics

Course objectives:

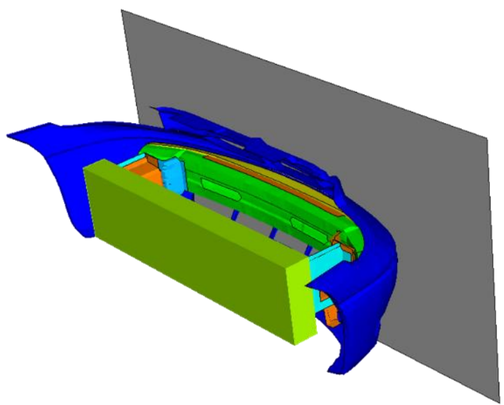
Upon completion of this course you will be able to:

- Use the explicit dynamics method effectively, including the application of general contact, mass scaling, and adaptive remeshing
- Use Abaqus/Explicit and Abaqus/Standard together to solve difficult problems, including results transfer and co-simulation
- Model high-strain-rate deformation and failure
- Filter output

Targeted audience: Simulation Analysts

Prerequisites: This course is recommended for engineers with experience using Abaqus

Training duration: **3 days**



1.5. Analysis of Composite Materials with Abaqus

Course objectives:

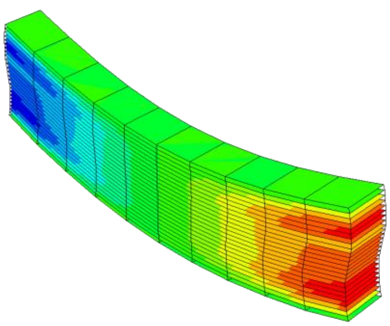
Upon completion of this course you will be able to:

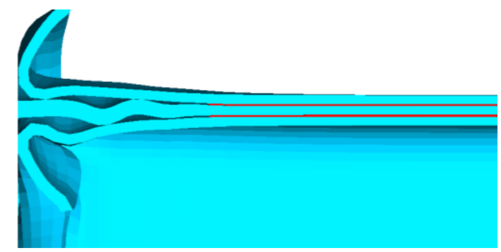
- Define anisotropic elasticity with Hookean models for combining the fiber-matrix response
- Define composite layups using Abaqus/CAE
- Model sandwich composite structures and stiffened composite panels
- Model progressive damage and failure in composites
- Model delamination and low-cycle fatigue of composite structures

Targeted audience: Simulation Analysts

Prerequisites: This course is recommended for engineers with experience using Abaqus/Standard

Training duration: **3 days**





1.6. Abaqus for Offshore Analysis

Course objectives:

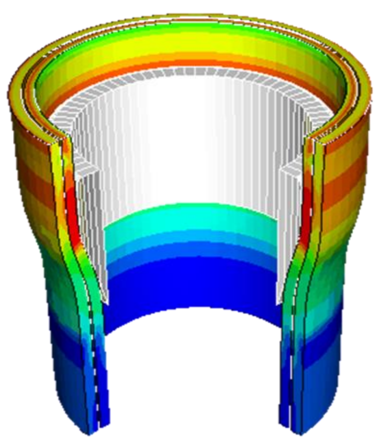
The topics covered in this course include:

- 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: This course is recommended for engineers with experience using Abaqus/Standard

Training duration: **2 days**



1.7. Analysis of Geotechnical Problems with Abaqus

Course objectives:

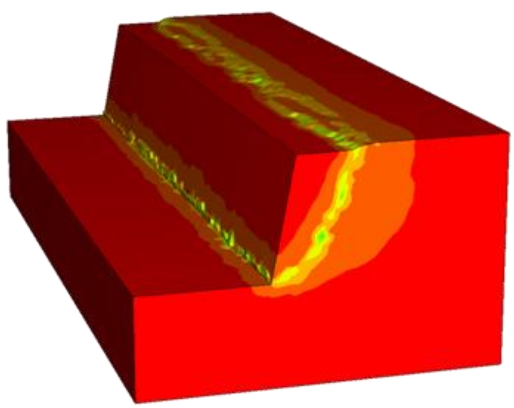
This course includes:

- An overview of modeling geotechnical problems
- Experimental testing and how it relates to the calibration of constitutive models for geotechnical materials
- How to use and calibrate the different geotechnical material constitutive models available in Abaqus
- The limitations of these models
- The coupling between fluid flow and stress/deformation in the analysis of porous media
- Modeling issues related to geotechnical problems

Targeted audience: This seminar is recommended for engineers with experience using Abaqus/Standard.

Prerequisites: This course is recommended for engineers with experience using Abaqus/Standard

Training duration: **2 days**



1.8. Writing User Subroutine with Abaqus

Course objectives:

In this course you will learn about:

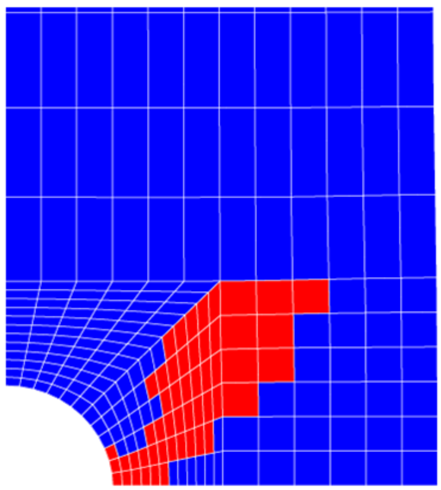
- When and how to use subroutines

- DLOAD, VDLOAD, and UTRACLOAD for specifying user-defined loading
- FILM for specifying user-defined film conditions
- USDFLD and VUSDFLD for defining field variable dependence
- UVARM for defining a user output variable
- UHYPER for modeling hyperelastic materials
- UMAT and VUMAT for allowing constitutive models to be added to the program
- UEL and VUEL for allowing the creation of user-defined elements

Targeted audience: This course is recommended for engineers with experience using Abaqus.

Prerequisites: A working knowledge of the finite element method and programming in either Fortran or C

Training duration: **2 days**



1.9. Python scripting in Abaqus

Course objectives

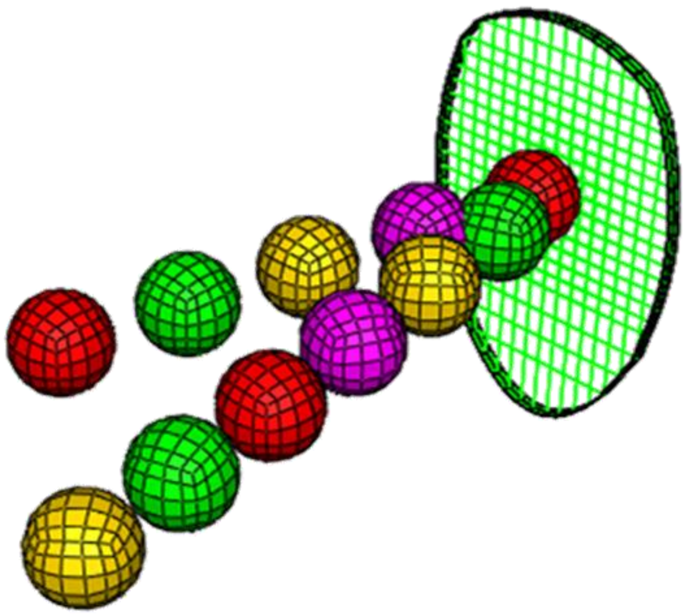
Upon completion of this course you will be able to:

- Help students to develop a high-level understanding of the Abaqus scripting capabilities.
- Organize and present the technical details of Python and the Abaqus Scripting Interface.
- Expose the strengths and weaknesses of Abaqus scripting.
- Encourage the student to use scripting in new ways.

Targeted audience: Simulation Analysts

Prerequisites: None

Training duration: 2 days



1.10. Advanced Abaqus Scripting

Course objectives

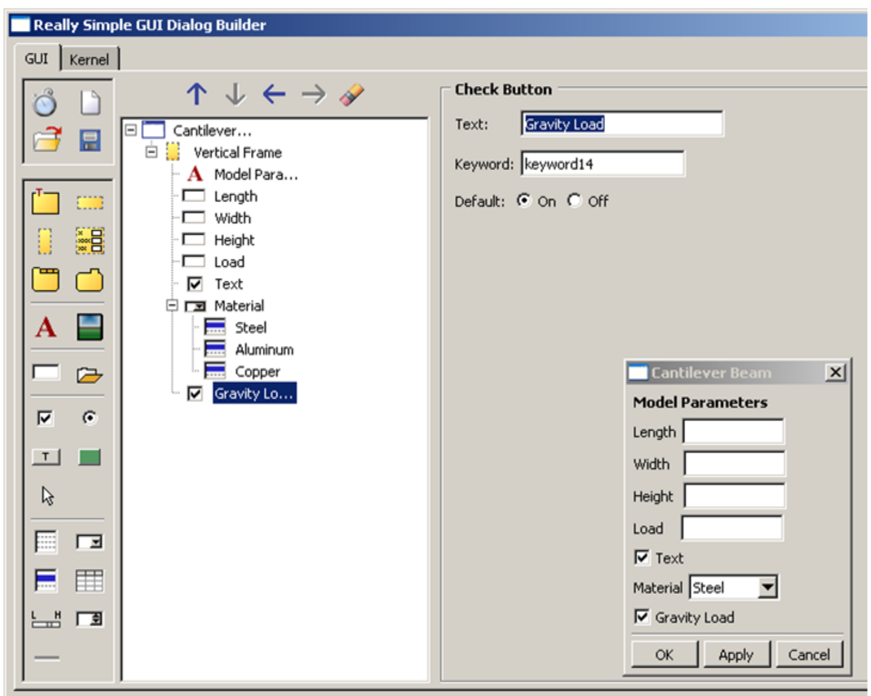
Upon completion of this course you will be able to:

- Help students to develop a high-level understanding of the Abaqus scripting capabilities and gain some proficiency.
- Organize and present the deeper technical details of Python and the Abaqus Scripting Interface.
- Expose the strengths and weaknesses of Abaqus scripting.
- Encourage the student to use scripting in new ways.

Targeted audience: Simulation Analysts

Prerequisites: Experience scripting with Python and Abaqus is recommended.

Training duration: **2 days**



1.11. Introduction to Isight

Course objectives

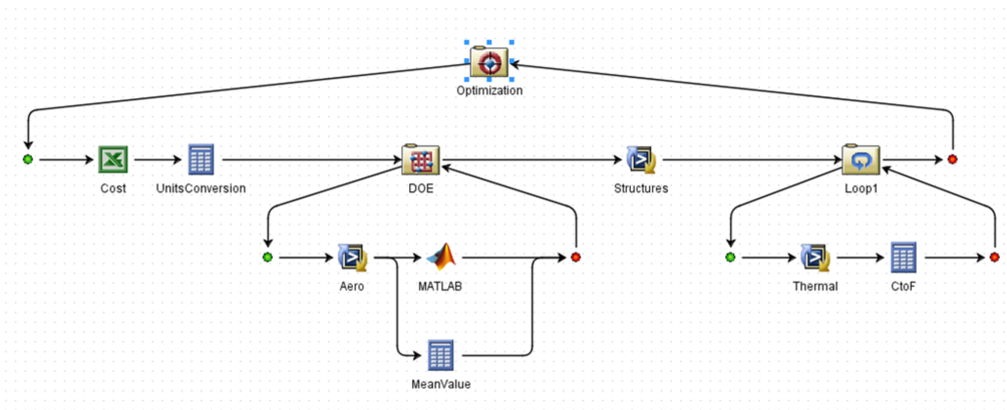
Upon completion of this course you will be able to:

- Automate a series of functions to create a Sim-flow
- Add components to a Sim-flow
- Set up the core component
- Configure components to pass data to/from each other
- Execute a Sim-flow
- Visualize Sim-flow results
- Evaluate design alternatives
- Create a Sim-flow to capture a process, by integrating various software in the company.
- Perform Design Optimization and gain Design Space understanding by using various techniques such as DOE, Optimization, Monte Carlo etc.

Targeted audience: Simulation Analysts

Prerequisites: None

Training duration: **2 days**



1.12. Introduction to fe-safe:

Course objectives:

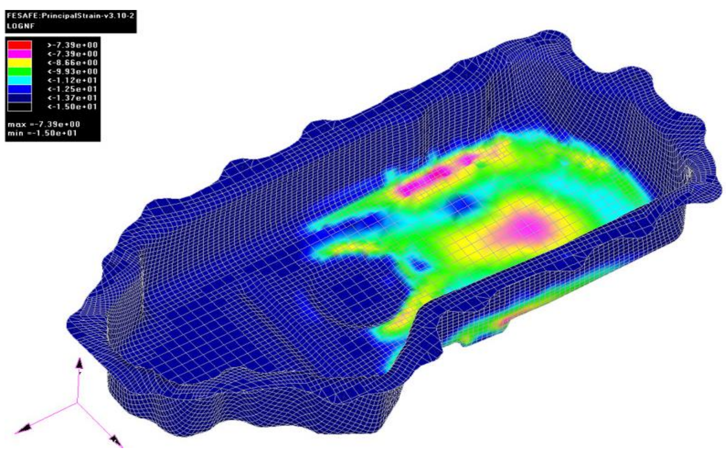
Upon completion of this course you will be able to set up and run various fatigue analyses using *fe-safe*

- Set up models and import models into *fe-safe*
- Select a material for fatigue analysis
- Set up your loadings
- Run various analyses in *fe-safe*

Targeted audience: Simulation Analysts

Prerequisites: None

Training duration: **2 days**



1.13. Non-parametric optimization with Abaqus and Tosca:

Course objectives:

Upon completion of this course you will be able to:

- Apply topology, shape, sizing and bead optimization techniques to your designs and produce lightweight, strong and durable components
- Reduce iterations of designs
- Use the optimization interface in Abaqus/CAE and Tosca/Structure for setup, execution, monitoring and postprocessing of topology, shape, sizing and bead optimization problems
- Use Tosca's structural optimization capability for highly nonlinear problems

Targeted audience: Simulation Analysts, Tosca Structure users

Prerequisites: Basic knowledge of Tosca Structure and Abaqus/CAE

Training duration: **2 day**