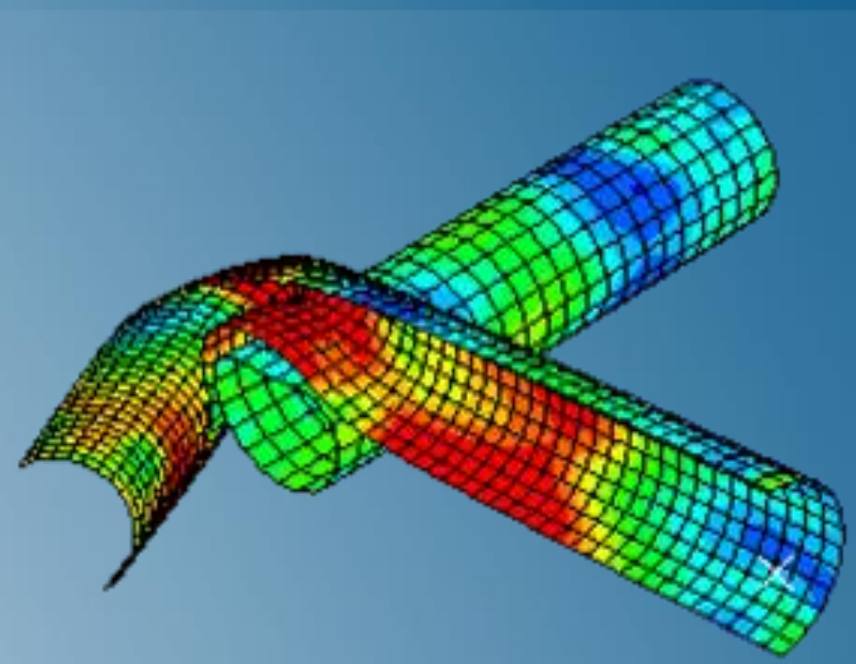
Introduction to Abaqus/Standard and Abaqus/Explicit

Abaqus 2022



3DEXPERIENCE®

About this Course

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

None

3 days

Day 1

Lesson 1	Defining an Abaqus Model
Workshop 1	Basic Input and Output
Lesson 2	Linear Static Analysis
Workshop 2	Linear Static Analysis of a Cantilever Beam: Multiple Load Cases
Lesson 3	Nonlinear Analysis in Abaqus/Standard
Workshop 3	Nonlinear Statics

Day 2

Lesson 4	Multistep Analysis in Abaqus
Workshop 4	Unloading Analysis
Lesson 5	Constraints and Contact
Workshop 5	Seal Contact
Lesson 6	Introduction to Dynamics
Workshop 6	Dynamics

Day 3

Lesson 7	Using Abaqus/Explicit
Workshop 7	Contact with Abaqus/Explicit
Lesson 8	Quasi-Static Analysis in Abaqus/Explicit
Workshop 8	Quasi-Static Analysis (*Optional*)
Lesson 9	Combining Abaqus/Standard & Abaqus/Explicit
Workshop 9	Import Analysis (*Optional*)

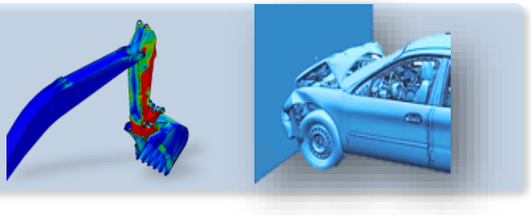
Additional Material

Appendix 1	Element Selection Criteria
Appendix 2	Material and Section Properties
Appendix 3	Contact Issues Specific to Abaqus/Standard
Appendix 4	Contact Issues Specific to Abaqus/Explicit

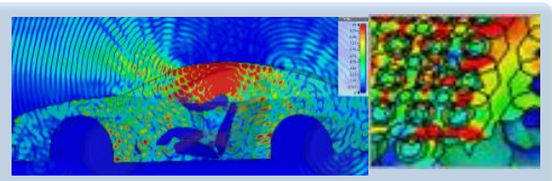
SIMULIA

- 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.

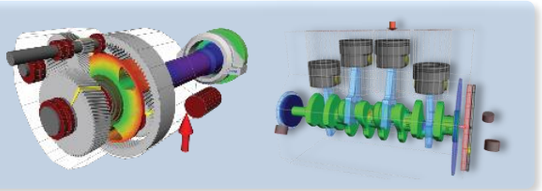
Structures



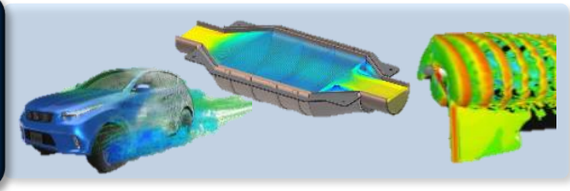
Electromagnetics



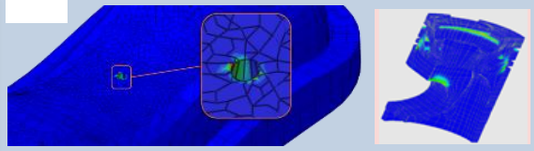
Multibody



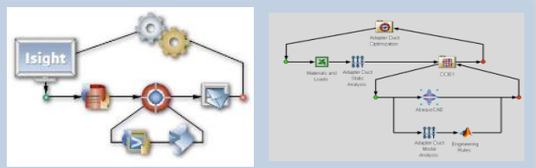
Fluids



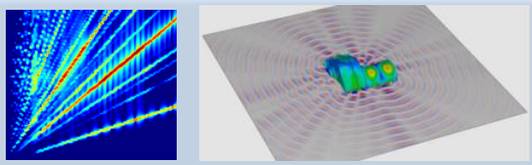
Durability



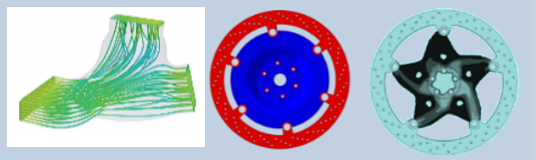
Automation



Vibro-acoustics



Optimization

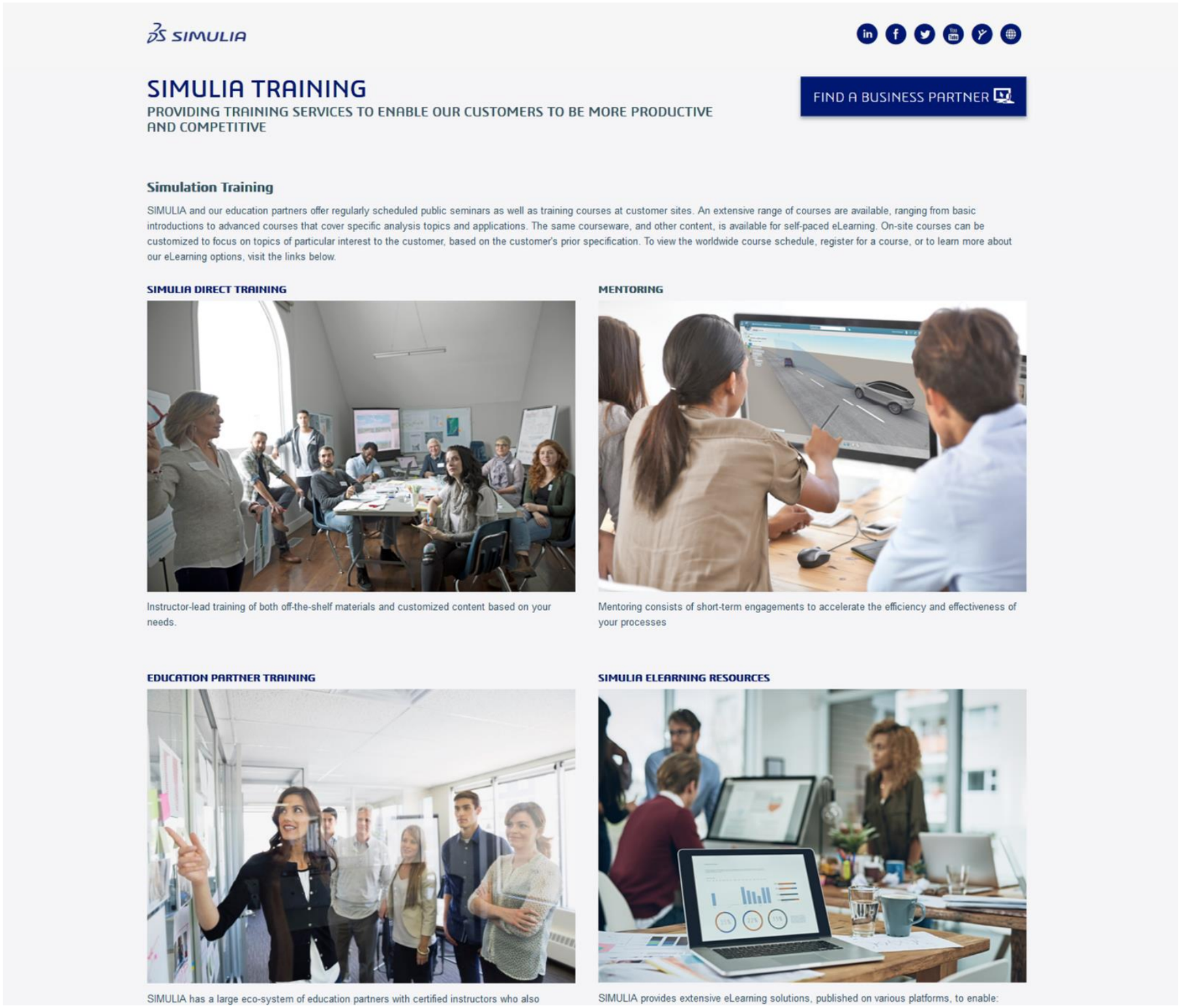


Join the Community!

Go to go.3ds.com/simc to log in or join!

SIMULIA Training

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



SIMULIA

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

Revision Status

Lesson 1	11/21	Updated for Abaqus 2022
Lesson 2	11/21	Updated for Abaqus 2022
Lesson 3	11/21	Updated for Abaqus 2022
Lesson 4	11/21	Updated for Abaqus 2022
Lesson 5	11/21	Updated for Abaqus 2022
Lesson 6	11/21	Updated for Abaqus 2022
Lesson 7	11/21	Updated for Abaqus 2022
Lesson 8	11/21	Updated for Abaqus 2022
Lesson 9	11/21	Updated for Abaqus 2022
Appendix 1	11/21	Updated for Abaqus 2022
Appendix 2	11/21	New for Abaqus 2022
Appendix 3	11/21	Updated for Abaqus 2022
Appendix 3	11/21	Updated for Abaqus 2022

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

Lesson 1: Defining an Abaqus Model

Lesson content:

- Introduction
- Abaqus FEA
- Abaqus/CAE
- Abaqus/Standard and Abaqus/Explicit
- Documentation
- SIMULIA Community
- Components of an Abaqus Model
- Details of an Abaqus Input File
- Abaqus Conventions
- Output
- Example: Cantilever Beam Model
- Parts and Assemblies (optional)
- Abaqus Environment Settings
- Managing Memory and Disk Resources
- Abaqus Fetch Utility
- Workshop Preliminaries
- Workshop 1: Basic Input and Output (IA)
- Workshop 1: Basic Input and Output (KW)

Both interactive (IA) and keywords (KW) versions of the workshop are provided. Complete only one.

2 hours

Lesson 2: Linear Static Analysis

Lesson content:

- Linear and Nonlinear Procedures
- Linear Static Analysis and Multiple Load Cases
- Multiple Load Case Usage
- Examples
- Workshop 2: Linear Static Analysis of a Cantilever Beam (IA)
- Workshop 2: Linear Static Analysis of a Cantilever Beam (KW)

Both interactive (IA) and keywords (KW) versions of the workshop are provided. Complete only one.

2 hours

Lesson 3: Nonlinear Analysis in Abaqus

Lesson content:

- Nonlinearity in Structural Mechanics
- Equations of Motion
- Nonlinear Analysis Using Implicit Methods
- Nonlinear Analysis Using Explicit Methods
- Input File for Nonlinear Analysis
- Status File
- Message File
- Output from Nonlinear Cantilever Beam Analysis
- Workshop 3: Nonlinear Statics (IA)
- Workshop 3: Nonlinear Statics (KW)

Both interactive (IA) and keywords (KW) versions of the workshop are provided. Complete only one.

2 hours

Lesson 4: Multistep Analysis in Abaqus

Lesson content:

- Multistep Analyses
- Restart Analysis in Abaqus
- Postprocessing of Restart Data
- Workshop 4: Unloading Analysis (IA)
- Workshop 4: Unloading Analysis (KW)

Both interactive (IA) and keywords (KW) versions of the workshop are provided. Complete only one.

1 hour

Lesson 5: Constraints and Contact

Lesson content:

- Constraints
- Tie Constraints
- Rigid Bodies
- Shell-to-solid Coupling
- Contact
- Defining General Contact
- Defining Contact Pairs
- Defining Surfaces
- Local Surface Behavior
- Relative Sliding of Points in Contact
- Adjusting Initial Nodal Locations for Contact
- Contact Output
- Workshop 5: Seal Contact (IA)
- Workshop 5: Seal Contact (KW)

Both interactive (IA) and keywords (KW) versions of the workshop are provided. Complete only one.

2.5 hours

Lesson 6: Introduction to Dynamics

Lesson content:

- What Makes a Problem Dynamic?
- Equations for Dynamic Problems
- Linear Dynamics
- Nonlinear Dynamics
- Comparing Abaqus/Standard and Abaqus/Explicit
- Nonlinear Dynamics Example
- Workshop 6: Dynamics (IA)
- Workshop 6: Dynamics (KW)

Both interactive (IA) and keywords (KW) versions of the workshop are provided. Complete only one.

2 hours

Lesson 7: Using Abaqus/Explicit

Lesson content:

- Overview of the Explicit Dynamics Procedure
- Abaqus/Explicit Syntax
- Rigid Bodies
- Workshop 7: Contact with Abaqus/Explicit (IA)
- Workshop 7: Contact with Abaqus/Explicit (KW)

Both interactive (IA) and keywords (KW) versions of the workshop are provided. Complete only one.

2 hours

Lesson 8: Quasi-Static Analysis in Abaqus/Explicit

Lesson content:

- Introduction
- Solution Strategies
- Quasi-Static Simulations Using Explicit Dynamics
- Energy Balance
- Example: Load Rates
- Example: Mass Scaling
- Adaptive Meshing
- Workshop 8: Quasi-Static Analysis (IA)
- Workshop 8: Quasi-Static Analysis (KW)

Both interactive (IA) and keywords (KW) versions of the workshop are provided. Complete only one.

2 hours

Lesson 9: Combining Abaqus/Standard & Abaqus/Explicit

Lesson content:

- Introduction
- Abaqus Usage
- Springback Calculation using Abaqus/Standard
- Workshop 9: Import Analysis (IA)
- Workshop 9: Import Analysis (KW)

Both interactive (IA) and keywords (KW) versions of the workshop are provided. Complete only one.

1 hour

Appendix 1: Element Selection Criteria

Appendix content:

- Elements
- Structural (Shells and Beams) vs. Continuum Elements
- Modeling Bending Using Continuum Elements
- Stress Concentrations
- Contact
- Incompressible Materials
- Mesh Generation
- Solid Element Selection Summary

1.5 hours

Appendix 2: Material and Section Properties

Appendix content:

- Abaqus Material Definitions
- Abaqus Conventions
- Linear Elasticity
- Large Strain Elasticity
- Metal Plasticity
- Section Properties

45 minutes

Appendix 3: Contact Issues Specific to Abaqus/Standard

Appendix content:

- Contact as Part of the Model Definition
- Mesh Density Considerations
- Contact Logic in Abaqus/Standard

30 minutes

Appendix 4: Contact Issues Specific to Abaqus/Explicit

Appendix content:

- Contact Pairs as Part of the History Data
- Enforcing the Contact Constraints
- Double-Sided Contact
- Initial Kinematic Compliance

30 minutes