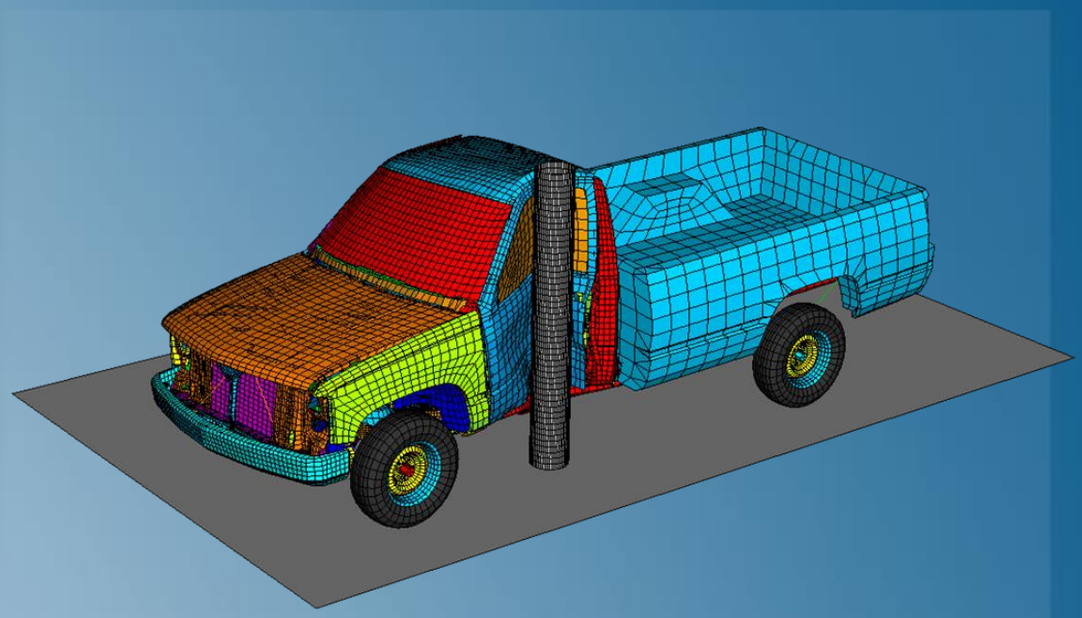
Crashworthiness Analysis with Abaqus

Abaqus 2022



3DEXPERIENCE®

About this Course

Course objectives

This course covers:

- Abaqus fundamentals and input syntax
- General "automatic" contact modeling
- Element selection for crash simulation
- Constraints and connections modeling
- Material models used in crash simulation
- Multiple mechanism damage and failure modeling

Targeted audience

New and experienced users of Abaqus who will perform structural crashworthiness or occupant safety simulations.

Prerequisites

No previous knowledge of Abaqus is required, but knowledge of finite elements and engineering mechanics is necessary.

3 days

Day 1

- Lesson 1 Introduction and Motivation
- Lesson 2 Setting up an Abaqus Model
- Lesson 3 Explicit Dynamics in Abaqus
- Lesson 4 Contact Modeling
  - Workshop 1 Impact of a Dodge Caravan Bumper against a Rigid Barrier

Day 2

- Lesson 5 Element Technology
- Lesson 6 Constraints and Connections
  - Workshop 2 Crash Analysis of a Rail
  - Workshop 3 Door Pole-Intrusion Test
  - Workshop 4 Iltis All-Terrain Vehicle Curb Strike
- Lesson 7 Material Modeling

Important note: Submit the global model for Workshop 7 prior to completing work on this day.

Day 3

- Lesson 8 Advanced Analysis Techniques
  - Workshop 5 Side Impact Analysis of a Pickup Truck using Submodeling Technique
- Lesson 9 Crash Output
  - Workshop 6 Curved Beam Analysis
- Lesson 10 Co-simulation
  - Workshop 7 Beam Impact Co-simulation

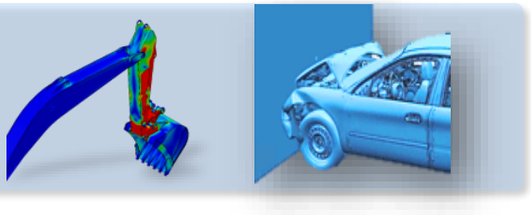
Additional Material

- Appendix 1 Contact Pairs
- Appendix 2 Seatbelts
  - Workshop 8 Seatbelt Safety System
- Appendix 3 Airbags
  - Workshop 9 Deployment of a Multi-Chambered Airbag
- Appendix 4 Tire Modeling and Analysis
- Appendix 5 Output Filtering
  - This appendix includes a detailed discussion of output filtering for general applications; however, the information is relevant for crash analysis.
- Appendix 6 Translators

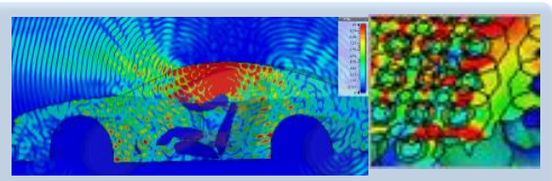
SIMULIA

- 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.

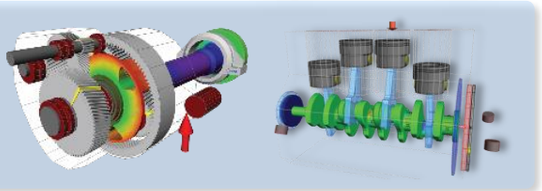
Structures



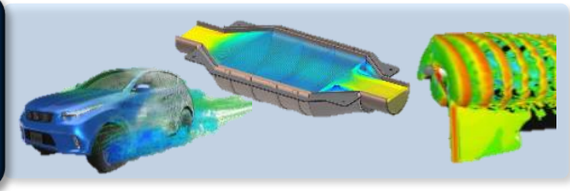
Electromagnetics



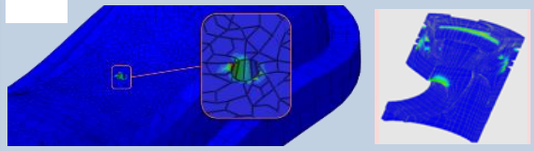
Multibody



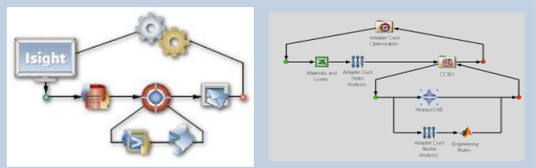
Fluids



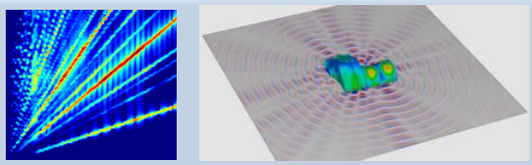
Durability



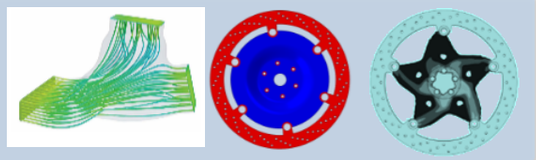
Automation



Vibro-acoustics



Optimization

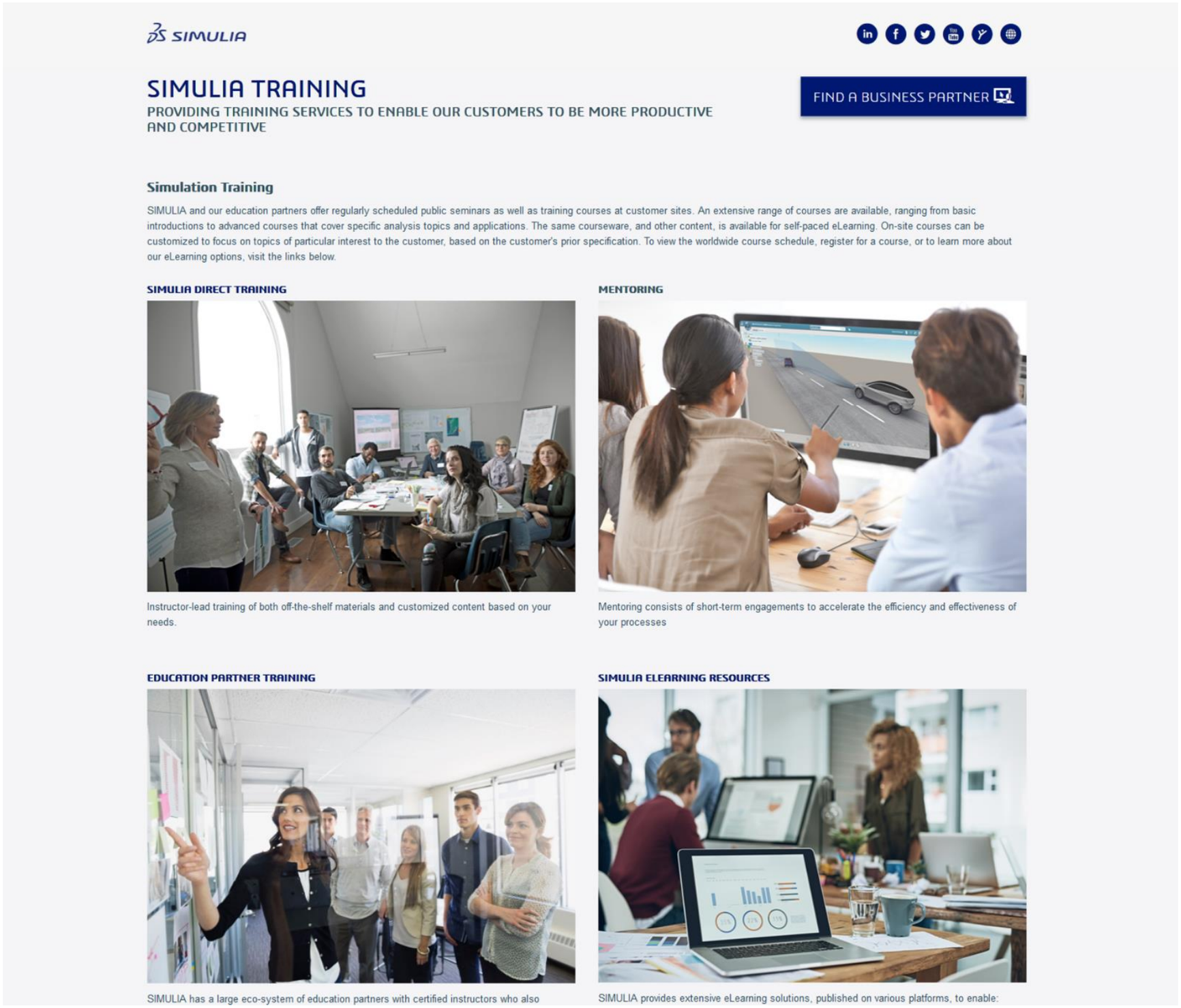


Join the Community!

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
Lesson 10	11/21	Updated for Abaqus 2022
Appendix 1	11/21	Updated for Abaqus 2022
Appendix 2	11/21	Updated for Abaqus 2022
Appendix 3	11/21	Updated for Abaqus 2022
Appendix 4	11/21	Updated for Abaqus 2022
Appendix 5	11/21	Updated for Abaqus 2022
Appendix 6	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 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: Introduction and Motivation

Lesson content:

- Background
- Selected Crashworthiness Applications
- Abaqus Crashworthiness Functionality

1 hour

Lesson 2: Setting up an Abaqus analysis

Lesson content:

- Components of an Abaqus Model
- Details of an Abaqus Input File
- Abaqus Input Conventions
- Abaqus Output
- Loads and Boundary Conditions
- Initial Conditions
- Example: Tube Crush Model
- Results Visualization
- Documentation
- Parallel Execution

2 hours

Lesson 3: Explicit Dynamics in Abaqus

Lesson content:

- What is Explicit Dynamics?
- Overview of Abaqus/Explicit
- Stable Time Increment
- Mass Scaling

1 hour

Lesson 4: Contact Modeling

Lesson content:

- Contact in Abaqus/Explicit
- Overview of General Contact
- Basic Features of General Contact
- Keyword Interface
- Additional Features of General Contact
- General Contact Output
- Tips for Diagnosing Contact Errors
- Additional Information
- Workshop Preliminaries
- Workshop 1: Impact of a Dodge Caravan Bumper against a Rigid Barrier

2 hours

Lesson 5: Element Technology

Lesson content:

- Introduction
- Designing the Crash Mesh
- Solid Elements
- Shell Elements
- Membrane Elements
- Beam and Truss Elements
- Special-Purpose Elements
- Section Controls to Modify Element Formulation

1.5 hours

Lesson 6: Constraints and Connections

Lesson content:

- Introduction
- Multi-Point Constraints
- Rigid Bodies
- Surface-Based Coupling Constraints
- Connector Elements
- Surface-Based Tie Constraints
- Offset Tied Interfaces
- Mesh-Independent Fasteners
- Cohesive Connections
- Tips for Diagnosing Constraint and Connection Errors
- Workshop 2: Crash Analysis of a Rail
- Workshop 3: Door Pole-Intrusion Test
- Workshop 4: Iltis All-Terrain Vehicle Curb Strike

3 hours

Lesson 7: Material Modeling

Lesson content:

- Introduction
- Material Data Definition
- Metal Plasticity
- Progressive Damage and Failure
- Hyperelastic Solid Rubbers
- Hyperfoam
- Low Density Foam
- Crushable Foams
- Other Material Properties and Models
- Encrypting Material Data

2 hours

Lesson 8: Advanced Analysis Techniques

Lesson content:

- Static Initialization and Import
- Selective Subcycling
- Submodeling
- Incorporating Manufacturing Effects
- Quasi-Static Analysis
- Restart
- Workshop 5: Side Impact Analysis of a Pickup Truck using the Submodeling Technique

2 hours

Lesson 9: Output

Lesson content:

- Output
- Workshop 6: Curved Beam Analysis

2 hours

Lesson 10: Co-simulation

Lesson content:

- Introduction
- Examples
- Co-simulation Modeling
- Postprocessing
- Substructuring
- Workshop 7: Beam Impact Co-simulation

2 hours

Appendix 1: Contact Pairs

Appendix content:

- Contact Pairs

0.5 hour

Appendix 2: Seatbelts

Appendix content:

- Seatbelts
- Workshop 8: Seatbelt Safety System

2.5 hours

Appendix 3: Airbags

Appendix content:

- Airbags Overview
- Uniform Pressure Method
- Lumped Kinetic Molecular Method
- Coupled Eulerian-Lagrangian Method
- Workshop 9: Deployment of a Multi-Chambered Airbag

2.5 hours

Appendix 4: Tire Modeling and Analysis

Appendix content:

- Tire Modeling and Analysis

0.5 hour

Appendix 5: Output Filtering

Appendix content:

- Introduction
- What is aliasing?
- Preventing aliasing
- On-demand output
- Abaqus/Viewer postprocessing filters
- Filter options
- Filter distortions
- References

1 hour

Appendix 6: Translators

Appendix content:

- Translator from PAM-CRASH to Abaqus
- Translator from RADIOSS to Abaqus
- Translator from LS-DYNA to Abaqus

45 minutes