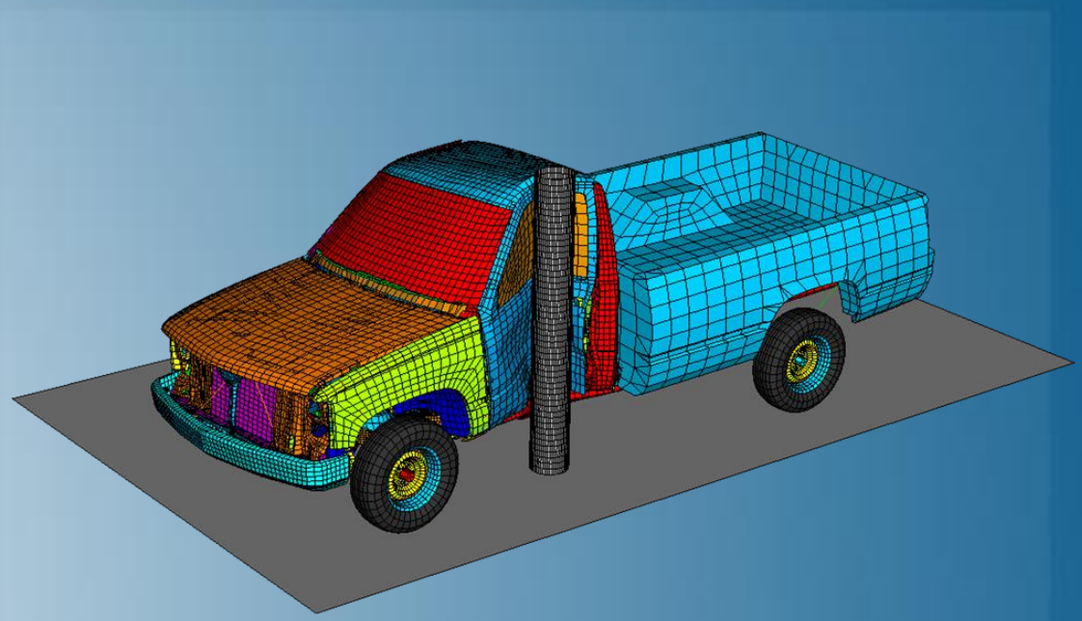
Crashworthiness Analysis with Abaqus

Abaqus 2020



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

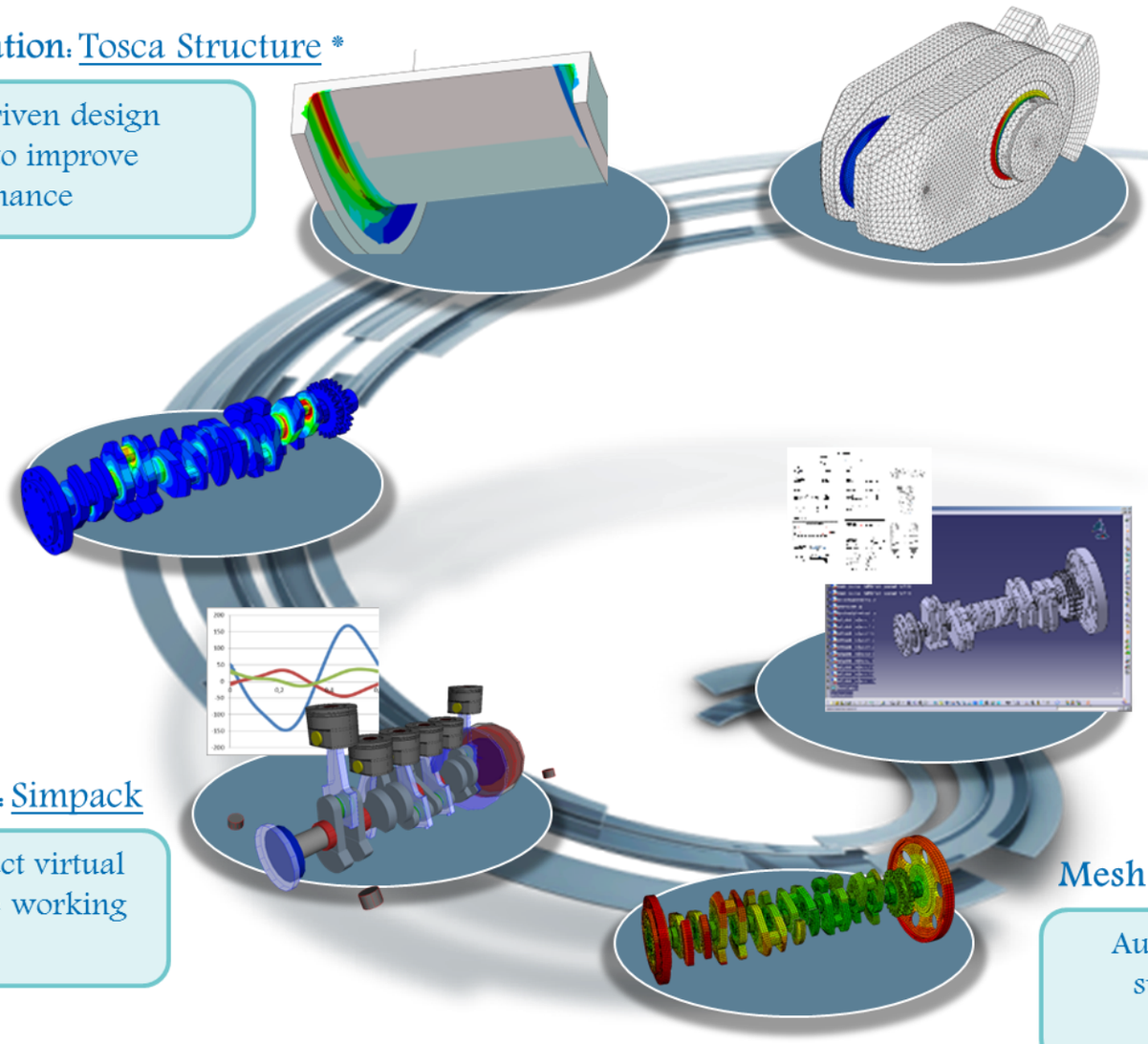
- SIMULIA is the Dassault Systèmes brand for Realistic Simulation solutions
- Portfolio of established, best-in-class products
  - Abaqus, Isight, Tosca, fe-safe, Simpack

Design Optimization: Tosca Structure *

Simulation-driven design refinement to improve performance

Durability Assessment: fe-safe *

Accurate life estimation to achieve certification



FEA Stress Analysis: Abaqus *

Detailed stress analysis using extracted load history from MBS

CAD Geometry: CATIA

Fully parameterized 3D geometry; FEA model generation via associative interface

Multibody Simulation: Simpack

System analysis to extract virtual load history of complete working cycle

Mesh Calibration: Isight *

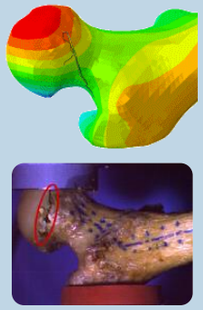
Automated mesh calibration: sufficient mesh quality for accurate results

* Included in extended licensing pool

SIMULIA's Power of the Portfolio

Abaqus

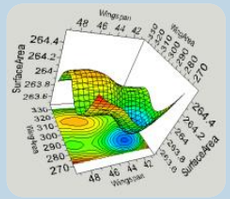
- Routine and Advanced Simulation
- Linear and Nonlinear, Static and Dynamic
- Thermal, Electrical, Acoustics
- Extended Physics through Co-simulation
- Model Preparation and Visualization



Realistic Human Simulation
High Speed Crash & Impact
Noise & Vibration

Isight

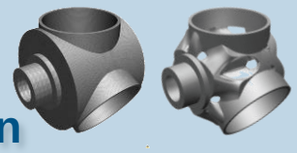
- Process Integration
- Design Optimization
- Parametric Optimization
- Six Sigma and Design of Experiments



Material Calibration
Workflow Automation
Design Exploration

Tosca

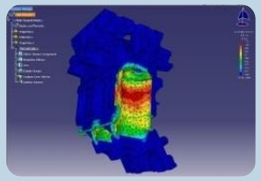
- Non-Parametric Optimization
- Structural and Fluid Flow Optimization
- Topology, Sizing, Shape, Bead Optimization



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

fe-safe

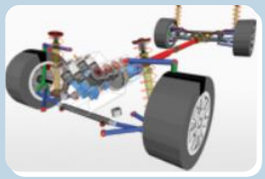
- Durability Simulation
- Low Cycle and High Cycle Fatigue
- Weld, High Temperature, Non-metallics



Safety Factors
Creep-Fatigue Interaction
Weld Fatigue

Simpack

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



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!

SIMULIA

Let the SIMULIA Learning Community be *Your* Portal to 21st Century Innovation

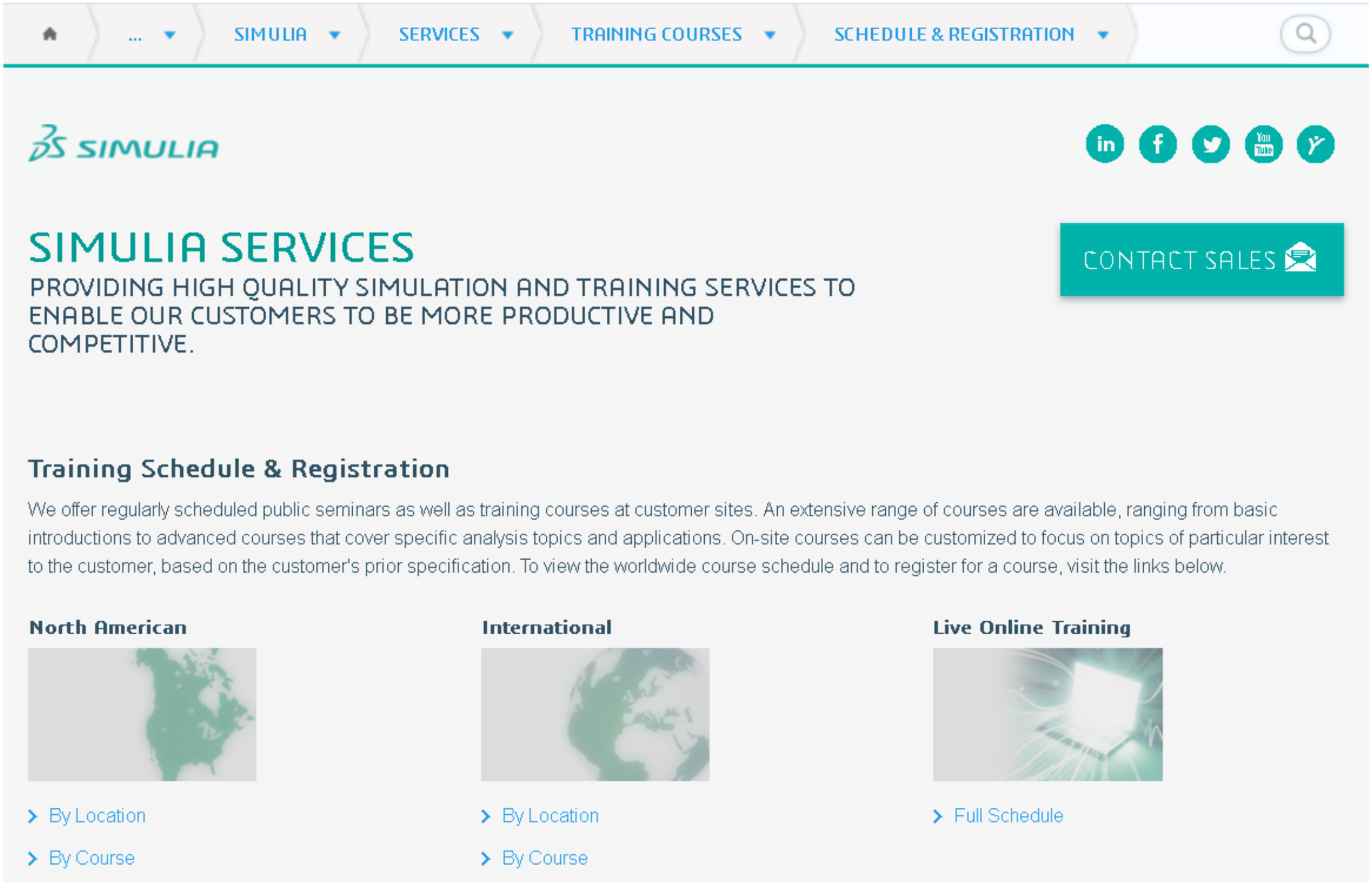
Discover new ways to explore how to leverage realistic simulation to drive product innovation. Join the thousands of Abaqus and Isight users who are already gaining valuable knowledge from the SIMULIA Learning Community.

For more information and registration, visit **3ds.com/simulia-learning**.
Connect. Share. Spark Innovation.

DASSAULT SYSTEMES | The **3D**EXPERIENCE Company

SIMULIA Training

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



Legal Notices

Revision Status

Lesson 1	11/19	Updated for Abaqus 2020
Lesson 2	11/19	Updated for Abaqus 2020
Lesson 3	11/19	Updated for Abaqus 2020
Lesson 4	11/19	Updated for Abaqus 2020
Lesson 5	11/19	Updated for Abaqus 2020
Lesson 6	11/19	Updated for Abaqus 2020
Lesson 7	11/19	Updated for Abaqus 2020
Lesson 8	11/19	Updated for Abaqus 2020
Lesson 9	11/19	Updated for Abaqus 2020
Lesson 10	11/19	Updated for Abaqus 2020
Appendix 1	11/19	Updated for Abaqus 2020
Appendix 2	11/19	Updated for Abaqus 2020
Appendix 3	11/19	Updated for Abaqus 2020
Appendix 4	11/19	Updated for Abaqus 2020
Appendix 5	11/19	Updated for Abaqus 2020
Appendix 6	11/19	Updated for Abaqus 2020

Workshop 1	11/19	Updated for Abaqus 2020
Workshop 2	11/19	Updated for Abaqus 2020
Workshop 3	11/19	Updated for Abaqus 2020
Workshop 4	11/19	Updated for Abaqus 2020
Workshop 5	11/19	Updated for Abaqus 2020
Workshop 7	11/19	Updated for Abaqus 2020
Workshop 8	11/19	Updated for Abaqus 2020
Workshop 9	11/19	Updated for Abaqus 2020

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
- 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