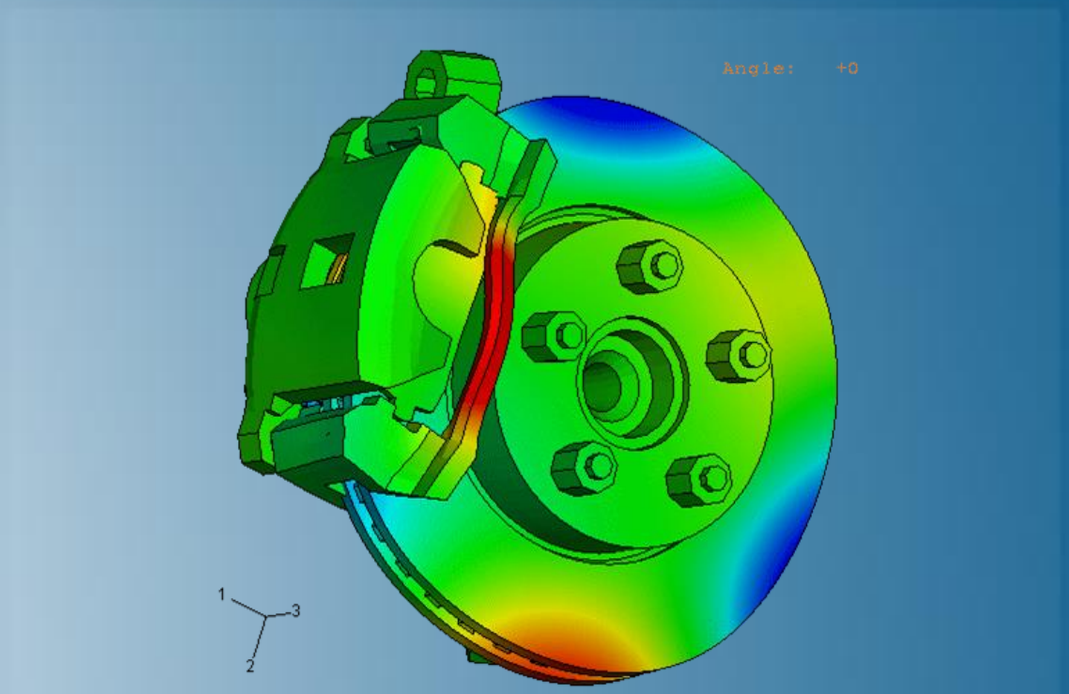
Automotive NVH with Abaqus

Abaqus 2018



3DEXPERIENCE®

About this Course

Course objectives

Upon completion of this course you will be able to:

- Perform natural frequency extractions
- Perform sound radiation analyses (acoustics)
- Include nonlinear preloading effects in your NVH simulations
- Perform Brake squeal analyses
- Create constraints and connections for Automotive NVH models
- Use substructuring techniques to run your NVH simulations more efficiently
- Perform advanced NVH postprocessing (via plug-ins)

Targeted audience

Simulation Analysts

Prerequisites

This course is recommended for engineers with experience using Abaqus

3 days

Day 1

- Lecture 1 Automotive NVH Overview
- Lecture 2 Modal Analysis
  - Workshop 1 Modal Analysis of a Control Arm
- Lecture 3 Steady-State Dynamics
  - Workshop 2 Steady State Dynamic Analysis of a Control Arm
- Lecture 4 Modal Transient Response

Day 2

- Lecture 5 Constraints and Interactions: Part 1
- Lecture 6 Constraints and Interactions: Part 2
  - Workshop 3 Constraints and Interactions for a Control Arm
- Lecture 7 Substructures
  - Workshop 4 Using Substructures to Model a Pick-up Truck
- Lecture 8 Base Motion Excitation
  - Workshop 5 Base Motion of a Pick-up Truck

Day 3

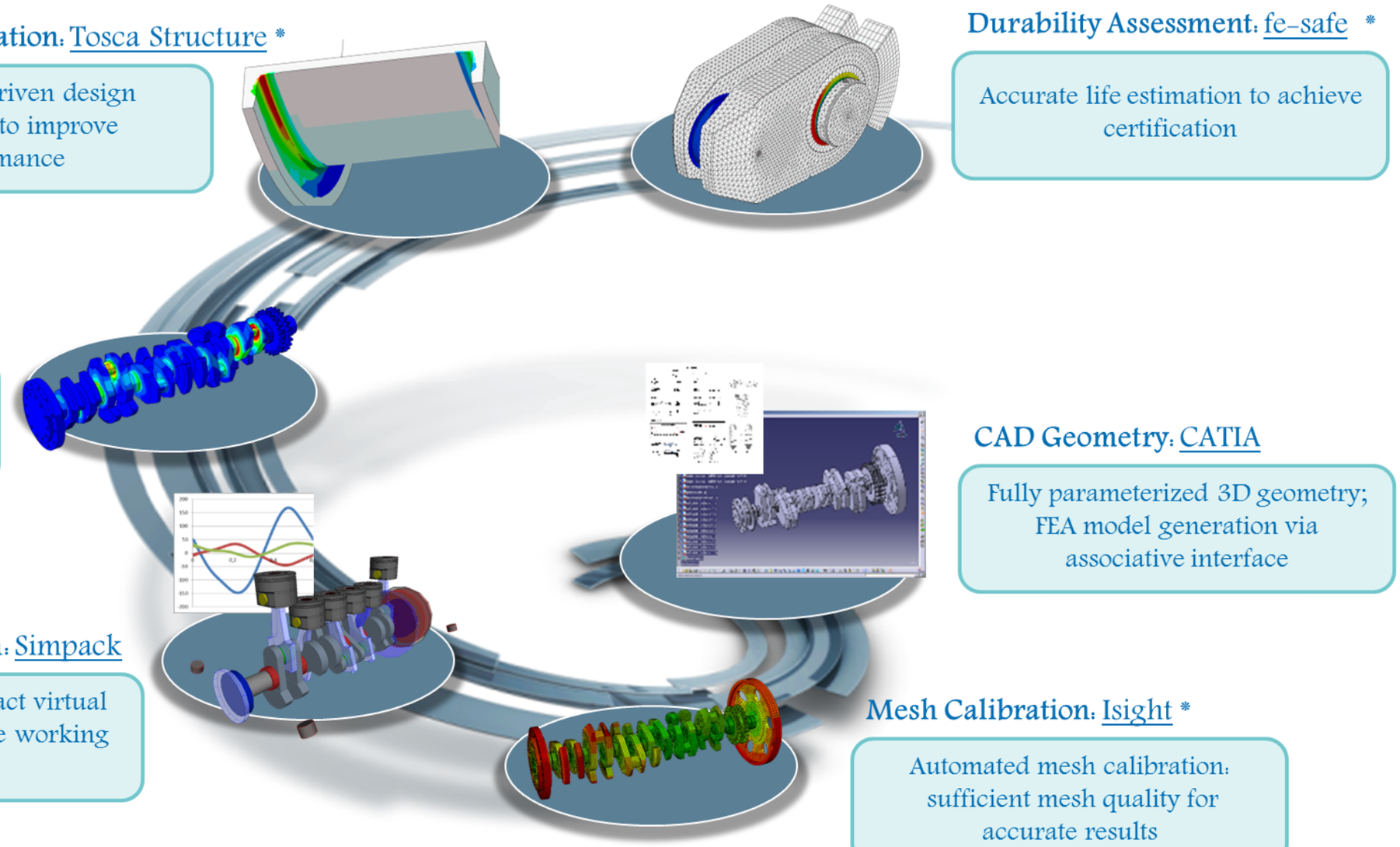
- Lecture 9 Coupled Structural-Acoustic Analysis
  - Workshop 6 Coupled Structural-Acoustic Analysis of a Truck
- Lecture 10 Brake Squeal Analysis
  - Workshop 7 Brake Squeal Analysis

Additional Material

- Appendix 1 Introduction to Modeling with Abaqus
- Appendix 2 Migrating from Nastran to Abaqus: Part 1
- Appendix 3 Migrating from Nastran to Abaqus: Part 2
  - Workshop 8 Nastran Translation: Control Arm Model
- Appendix 4 Abaqus-EXCITE Workflow

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 *

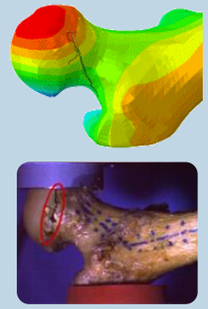
Automated mesh calibration: sufficient mesh quality for accurate results

* Included in extended licensing pool

SIMULIA's Power of the Portfolio

Abaqus

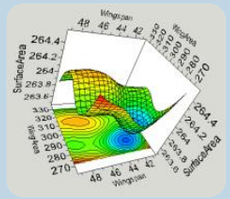
- Routine and Advanced Simulation
- Linear and Nonlinear, Static and Dynamic
- Thermal, Electrical, Acoustics
- Extended Physics through Co-simulation
- Model Preparation and Visualization



Realistic Human Simulation
High Speed Crash & Impact
Noise & Vibration

Isight

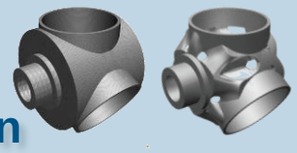
- Process Integration
- Design Optimization
- Parametric Optimization
- Six Sigma and Design of Experiments



Material Calibration
Workflow Automation
Design Exploration

Tosca

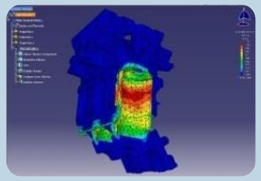
- Non-Parametric Optimization
- Structural and Fluid Flow Optimization
- Topology, Sizing, Shape, Bead Optimization



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

fe-safe

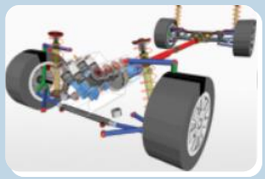
- Durability Simulation
- Low Cycle and High Cycle Fatigue
- Weld, High Temperature, Non-metallics



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

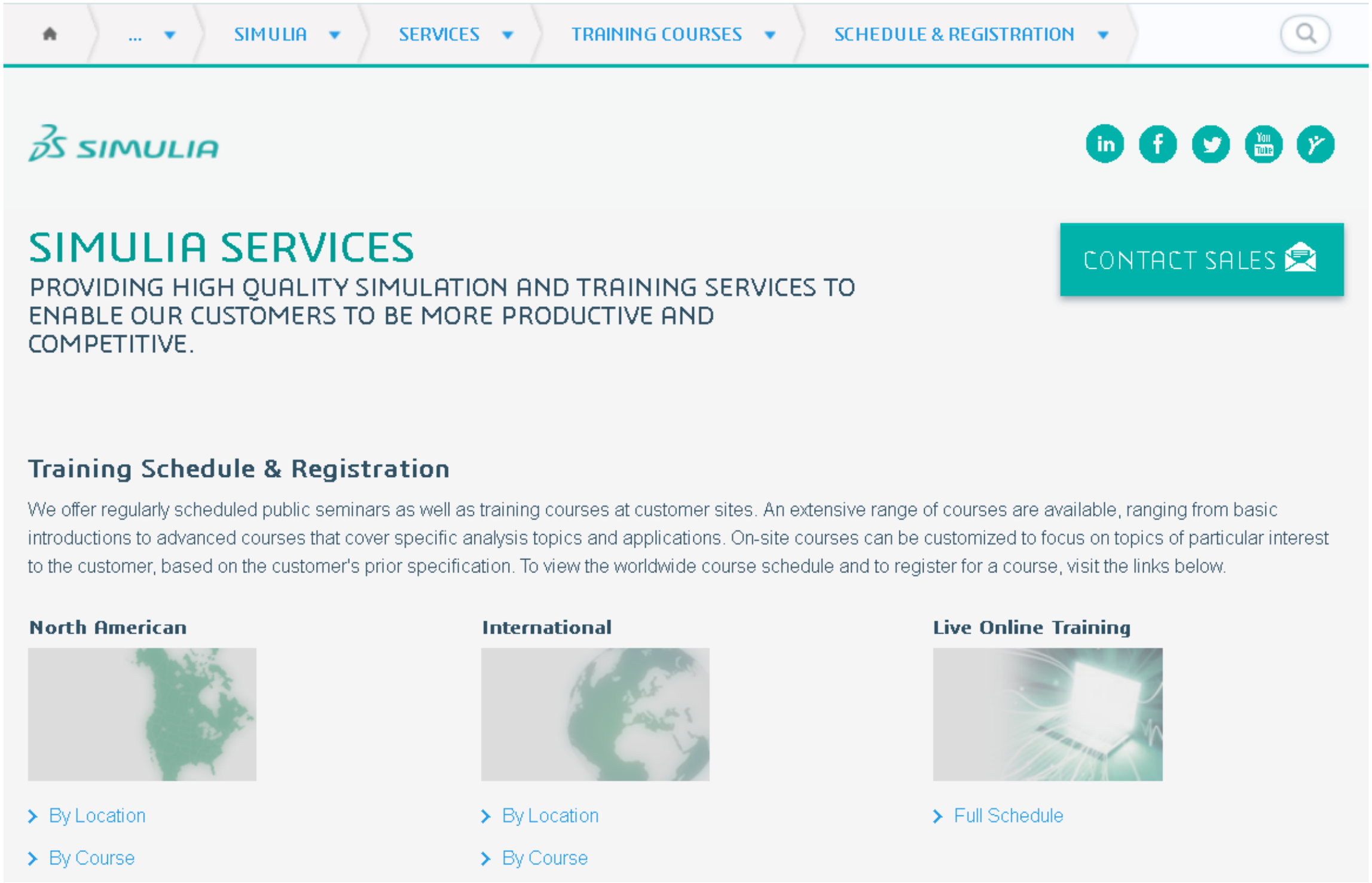
Discover new ways to explore how to leverage realistic simulation to drive product innovation. Join the thousands of Abaqus and Isight users who are already gaining valuable knowledge from the SIMULIA Learning Community.

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

Lecture 1	11/17	Updated for Abaqus 2018
Lecture 2	11/17	Updated for Abaqus 2018
Lecture 3	11/17	Updated for Abaqus 2018
Lecture 4	11/17	Updated for Abaqus 2018
Lecture 5	11/17	Updated for Abaqus 2018
Lecture 6	11/17	Updated for Abaqus 2018
Lecture 7	11/17	Updated for Abaqus 2018
Lecture 8	11/17	Updated for Abaqus 2018
Lecture 9	11/17	Updated for Abaqus 2018
Lecture 10	11/17	Updated for Abaqus 2018
Appendix 1	11/17	Updated for Abaqus 2018
Appendix 2	11/17	Updated for Abaqus 2018
Appendix 3	11/17	Updated for Abaqus 2018
Appendix 4	11/17	Updated for Abaqus 2018

Workshop 1	11/17	Updated for Abaqus 2018
Workshop 2	11/17	Updated for Abaqus 2018
Workshop 3	11/17	Updated for Abaqus 2018
Workshop 4	11/17	Updated for Abaqus 2018
Workshop 5	11/17	Updated for Abaqus 2018
Workshop 6	11/17	Updated for Abaqus 2018
Workshop 7	11/17	Updated for Abaqus 2018
Workshop 8	11/17	Updated for Abaqus 2018

Lesson 1: Automotive NVH Overview

Lesson content:

- Introduction
- From Component to Full Vehicle NVH
- Example Analyses
- Abaqus NVH Functionality
- Summary

1 hour

Lesson 2: Modal Analysis

Lesson content:

- Problem Formulation
- Eigenvalue Solution Methods
- Example: Engine Block Frequency Extraction
- Frequency Output
- Frequencies of Preloaded Structures
- Residual Modes
- Workshop Preliminaries
- Workshop 1: Modal Analysis of a Control Arm (IA)
- Workshop 1: Modal Analysis of a Control Arm (KW)

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

2.5 hours

Lesson 3: Steady-State Dynamics

Lesson content:

- Introduction
- Damping
- Damping Controls
- Steady-State Dynamics Solution Procedures
- Excitation and Output
- Mobility
- Steady-State Dynamics Usage Example
- Examples
- Workshop 2: Steady State Dynamic Analysis of a Control Arm (IA)
- Workshop 2: Steady State Dynamic Analysis of a Control Arm (KW)

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

2.5 hours

Lesson 4: Modal Transient Response

Lesson content:

- Introduction
- Excitation
- Output
- Examples

45 minutes

Lesson 5: Constraints and Interactions: Part 1

Lesson content:

- Introduction
- Rigid Bodies
- Surface-Based Coupling Constraints
- Surface-Based Tie Constraints
- Contact Interactions
- Automatic Contact Pair Detection

1 hour

Lesson 6: Constraints and Interactions: Part 2

Lesson content:

- Multi-Point Constraints
- Connector Elements
- Mesh-Independent Fasteners
- Workshop 3: Constraints and Interactions for a Control Arm (IA)
- Workshop 3: Constraints and Interactions for a Control Arm (KW)

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

2.5 hours

Lesson 7: Substructures

Lesson content:

- Introduction
- Substructure Modeling
- Preloading Substructures
- Dynamic Substructuring
- Substructure Output
- Substructuring Example: Rolling Tires
- Workshop 4: Using Substructures to Model a Pick-up Truck (IA)
- Workshop 4: Using Substructures to Model a Pick-up Truck (KW)

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

2.5 hours

Lesson 8: Base Motion Excitation

Lesson content:

- Introduction
- Primary Base Motion
- Secondary Base Motion
- Usage
- Example
- Workshop 5: Base Motion of a Pick-up Truck (IA)
- Workshop 5: Base Motion of a Pick-up Truck (KW)

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

2 hours

Lesson 9: Coupled Structural-Acoustic Analysis

Lesson content:

- Introduction
- Coupled Structural-Acoustics Modeling
- Analysis Procedures
- Damping
- Element Size
- Acoustic Infinite Elements
- Impedance
- Output
- Acoustic Contribution Factors
- Estimate Acoustic Radiation
- Workshop 6: Coupled Structural-Acoustic Analysis of a Truck (IA)
- Workshop 6: Coupled Structural-Acoustic Analysis of a Truck (KW)

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

2.5 hours

Lesson 10: Brake Squeal Analysis

Lesson content:

- Introduction
- Complex Eigenvalue Extraction
- Verifying Brake Squeal Simulations
- Examples
- Transient Dynamics
- References
- Workshop 7: Brake Squeal Analysis (IA)
- Workshop 7: Brake Squeal Analysis (KW)

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

2.5 hours

Appendix 1: Introduction to Modeling with Abaqus

Appendix content:

- Abaqus Conventions
- Details of an Abaqus Input File
- Overview of Abaqus/CAE
- Starting Abaqus/CAE
- Orphan Mesh Import
- Example

1 hour

Appendix 2: Migrating from Nastran to Abaqus: Part 1

Appendix content:

- Introduction
- Nastran and Abaqus Input Comparison
- Translator from Nastran to Abaqus
- Solution Procedure Translation
- Validating a Translated Model

75 minutes

Appendix 3: Migrating from Nastran to Abaqus: Part 2

Appendix content:

- Modeling Differences Between Abaqus and Nastran
- Element Differences Between Abaqus and Nastran
- Interface Differences Between Abaqus and Nastran
- Translation Troubleshooting
- Workshop 8: Nastran Translation: Control Arm Model (IA)
- Workshop 8: Nastran Translation: Control Arm Model (KW)

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

2 hours

Appendix 4: Abaqus-EXCITE Workflow

Appendix content:

- Introduction
- Abaqus-EXCITE Workflow
- Abaqus-EXCITE-Abaqus Workflow

45 minutes