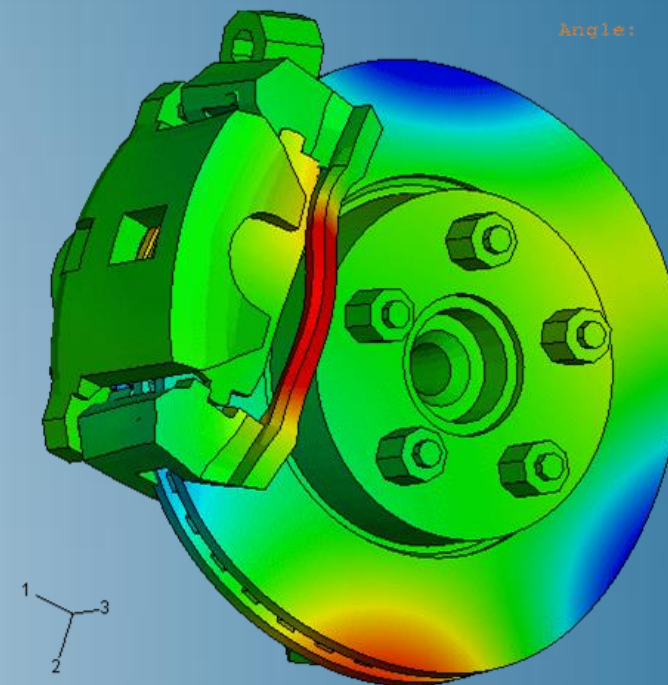


Automotive NVH with Abaqus

Abaqus 2019



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

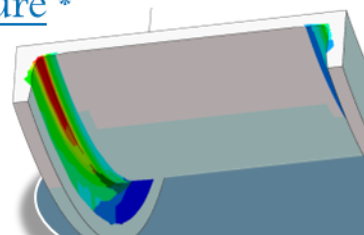
- ▶ Appendix 1 Introduction to Modeling with Abaqus
- ▶ Appendix 2 Migrating from Nastran to Abaqus: Part 1
- ▶ Appendix 3 Migrating from Nastran to Abaqus: Part 2
 - ▣ Workshop 8 Nastran Translation: Control Arm Model
- ▶ Appendix 4 Abaqus-EXCITE Workflow

SIMULIA

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

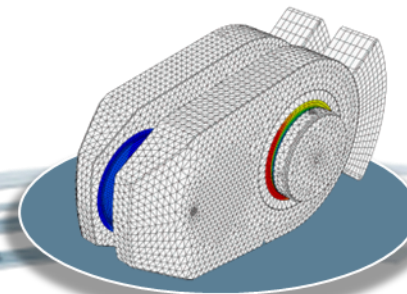
Design Optimization: Tosca Structure *

Simulation-driven design refinement to improve performance



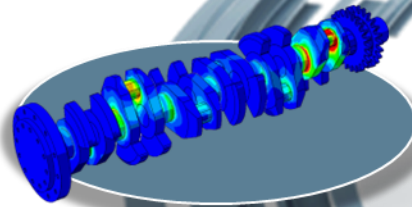
Durability Assessment: fe-safe *

Accurate life estimation to achieve certification



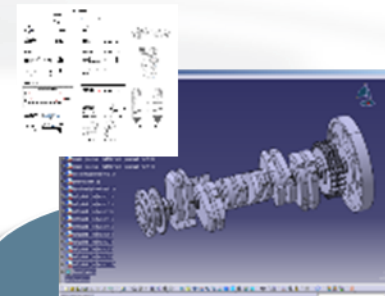
FEA Stress Analysis: Abaqus *

Detailed stress analysis using extracted load history from MBS



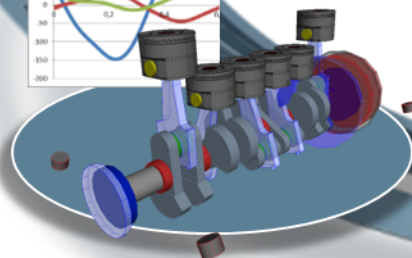
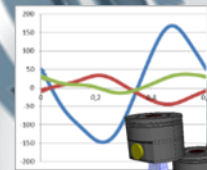
CAD Geometry: CATIA

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



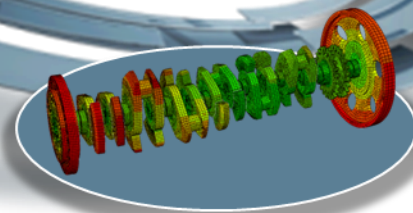
Multibody Simulation: Simpack

System analysis to extract virtual load history of complete working cycle



Mesh Calibration: Isight *

Automated mesh calibration; sufficient mesh quality for accurate results

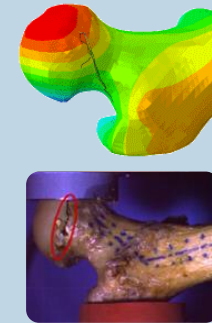


* Included in extended licensing pool

SIMULIA's Power of the Portfolio

Abaqus

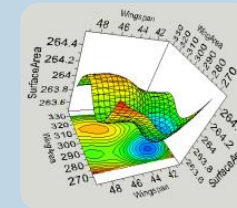
- Routine and Advanced Simulation
- Linear and Nonlinear, Static and Dynamic
- Thermal, Electrical, Acoustics
- Extended Physics through Co-simulation
- Model Preparation and Visualization



**Realistic Human Simulation
High Speed Crash & Impact
Noise & Vibration**

Isight

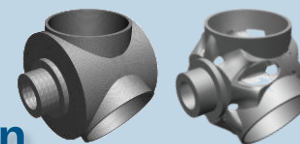
- Process Integration
- Design Optimization
- Parametric Optimization
- Six Sigma and Design of Experiments



**Material Calibration
Workflow Automation
Design Exploration**

Tosca

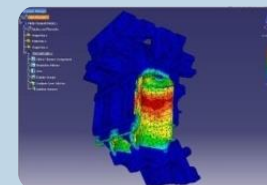
- Non-Parametric Optimization
- Structural and Fluid Flow Optimization
- Topology, Sizing, Shape, Bead Optimization



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

fe-safe

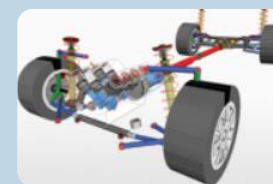
- Durability Simulation
- Low Cycle and High Cycle Fatigue
- Weld, High Temperature, Non-metallics



**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 **3DEXPERIENCE** Company

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

**SIMULIA****SERVICES****TRAINING COURSES****SCHEDULE & REGISTRATION**



SIMULIA SERVICES

PROVIDING HIGH QUALITY SIMULATION AND TRAINING SERVICES TO
ENABLE OUR CUSTOMERS TO BE MORE PRODUCTIVE AND
COMPETITIVE.

[CONTACT SALES](#) 

Training Schedule & Registration

We 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. 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 and to register for a course, visit the links below.

North American



- > By Location
- > By Course

International



- > By Location
- > By Course

Live Online Training



- > Full Schedule

Legal Notices

The software described in this documentation is available only under license from Dassault Systèmes or its subsidiaries and may be used or reproduced only in accordance with the terms of such license.

This documentation and the software described in this documentation are subject to change without prior notice.

Dassault Systèmes and its subsidiaries shall not be responsible for the consequences of any errors or omissions that may appear in this documentation.

No part of this documentation may be reproduced or distributed in any form without prior written permission of Dassault Systèmes or its subsidiaries.

© Dassault Systèmes, 2018

Printed in the United States of America.

Abaqus, the 3DS logo, and SIMULIA are trademarks or registered trademarks of Dassault Systèmes or its subsidiaries in the US and/or other countries.

Other company, product, and service names may be trademarks or service marks of their respective owners. For additional information concerning trademarks, copyrights, and licenses, see the Legal Notices in the SIMULIA User Assistance.

Revision Status

Lecture 1	11/18	Updated for Abaqus 2019
Lecture 2	11/18	Updated for Abaqus 2019
Lecture 3	11/18	Updated for Abaqus 2019
Lecture 4	11/18	Updated for Abaqus 2019
Lecture 5	11/18	Updated for Abaqus 2019
Lecture 6	11/18	Updated for Abaqus 2019
Lecture 7	11/18	Updated for Abaqus 2019
Lecture 8	11/18	Updated for Abaqus 2019
Lecture 9	11/18	Updated for Abaqus 2019
Lecture 10	11/18	Updated for Abaqus 2019
Appendix 1	11/18	Updated for Abaqus 2019
Appendix 2	11/18	Updated for Abaqus 2019
Appendix 3	11/18	Updated for Abaqus 2019
Appendix 4	11/18	Updated for Abaqus 2019

Workshop 1	11/18	Updated for Abaqus 2019
Workshop 2	11/18	Updated for Abaqus 2019
Workshop 3	11/18	Updated for Abaqus 2019
Workshop 4	11/18	Updated for Abaqus 2019
Workshop 5	11/18	Updated for Abaqus 2019
Workshop 6	11/18	Updated for Abaqus 2019
Workshop 7	11/18	Updated for Abaqus 2019
Workshop 8	11/18	Updated for Abaqus 2019

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