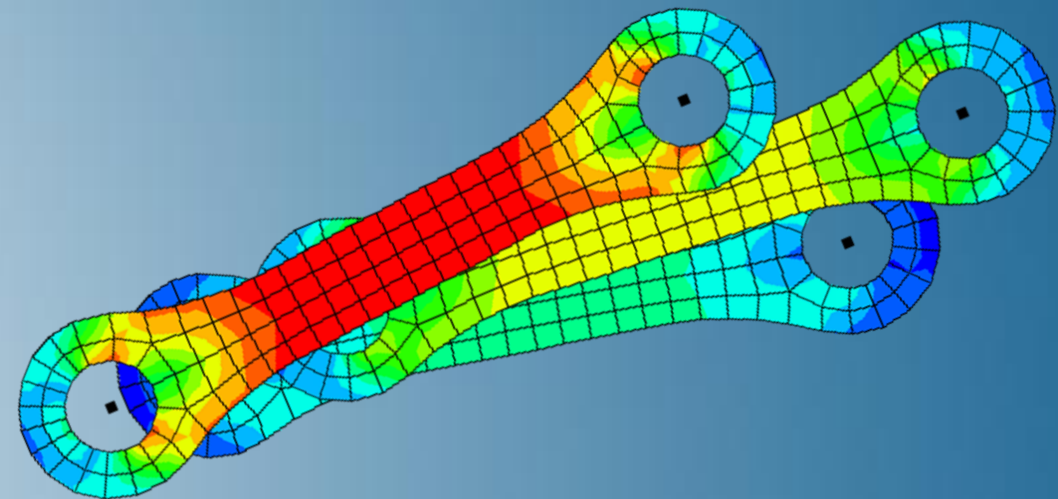


# Substructures and Submodeling with Abaqus

Abaqus 2018



**3DEXPERIENCE®**



# About this Course

## Course objectives

Upon completion of this course you will be able to:

- ▶ Understand the difference between substructuring and submodeling
- ▶ Build, translate, rotate and reflect substructures
- ▶ Build preloads into substructures
- ▶ Design meshes for submodel analysis
- ▶ Perform solid-to-solid, shell-to-shell, and shell-to-solid submodeling

## Targeted audience

Simulation Analysts

## Prerequisites

This course is recommended for engineers with experience using Abaqus



2 days

# Day 1

---

- ▶ Lecture 1            Introduction to Substructures
- ▶ Lecture 2            Using Static Substructuring in Abaqus
- ▶ Lecture 3            Linear Perturbations about a Preloaded State
- ▶ Lecture 4            Dynamic Substructuring
- ▶ Lecture 5            Substructure Output
- ▶ Lecture 6            Substructuring Examples
  - Workshop 1a Substructures: Plane Frame Analysis
  - Workshop 1b Substructures: Surface Mount Analysis
- ▶ Lecture 7            Using substructures with Abaqus/Explicit
  - Workshop 2   Substructures: Beam Impact (*optional*)

## Day 2

---

- ▶ Lecture 8            Introduction to Submodeling
- ▶ Lecture 9            Submodeling in Abaqus
- ▶ Lecture 10           Abaqus Usage and Examples (Part 1)
  - Workshop 3   Submodeling: Pressure Vessel Nozzle Analysis
- ▶ Lecture 11           Abaqus Usage and Examples (Part 2)
  - Workshop 4   Submodeling: Ceramic-Metal Braze Joint
- ▶ Lecture 12           Submodeling Practices
  - Workshop 5   Submodeling: Composite Tube Joint
- ▶ Lecture 13           Limitations of Submodeling

## Additional Material

---

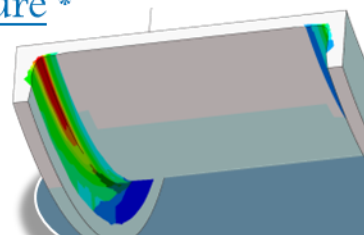
- ▶ Appendix 1      Theory of Substructures

# SIMULIA

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

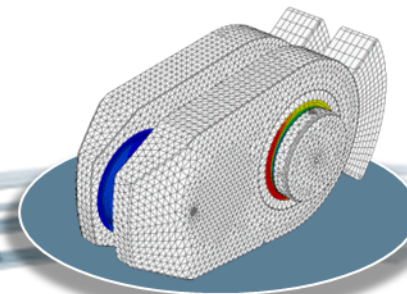
## Design Optimization: Tosca Structure \*

Simulation-driven design refinement to improve performance



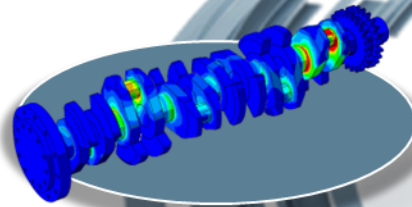
## Durability Assessment: fe-safe \*

Accurate life estimation to achieve certification



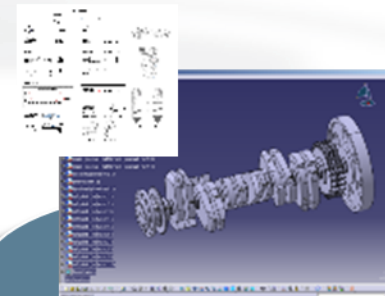
## FEA Stress Analysis: Abaqus \*

Detailed stress analysis using extracted load history from MBS



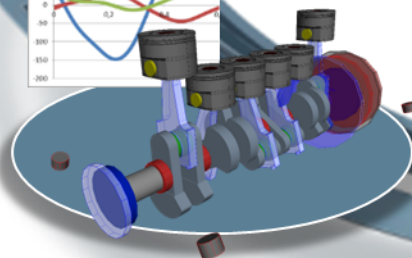
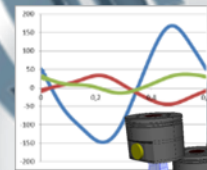
## CAD Geometry: CATIA

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



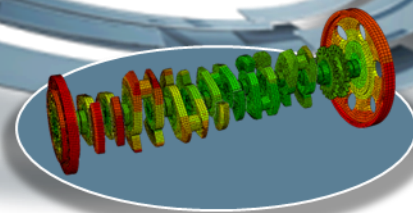
## Multibody Simulation: Simpack

System analysis to extract virtual load history of complete working cycle



## Mesh Calibration: Isight \*

Automated mesh calibration; sufficient mesh quality for accurate results



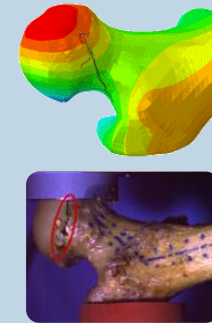
\* Included in extended licensing pool



# SIMULIA's Power of the Portfolio

## Abaqus

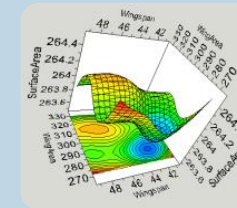
- Routine and Advanced Simulation
- Linear and Nonlinear, Static and Dynamic
- Thermal, Electrical, Acoustics
- Extended Physics through Co-simulation
- Model Preparation and Visualization



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

## Isight

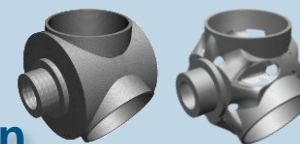
- Process Integration
- Design Optimization
- Parametric Optimization
- Six Sigma and Design of Experiments



**Material Calibration  
Workflow Automation  
Design Exploration**

## Tosca

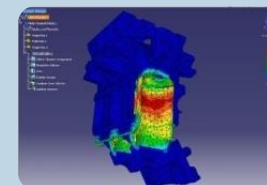
- Non-Parametric Optimization
- Structural and Fluid Flow Optimization
- Topology, Sizing, Shape, Bead Optimization



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

## fe-safe

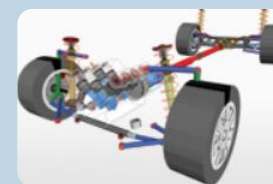
- Durability Simulation
- Low Cycle and High Cycle Fatigue
- Weld, High Temperature, Non-metallics



**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](http://www.3ds.com/slc)  
to log in or join!



 **SIMULIA**

Let the **SIMULIA Learning Community** be *Your* Portal to 21<sup>st</sup> 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](http://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, 2017

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	11/17	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
Lecture 11	11/17	Updated for Abaqus 2018
Lecture 12	11/17	Updated for Abaqus 2018
Lecture 13	11/17	Updated for Abaqus 2018
Appendix 1	11/17	Updated for Abaqus 2018

Workshop 1a	11/17	Updated for Abaqus 2018
Workshop 1b	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

# Lesson 1: Introduction to Substructures

## *Lesson content:*

- ▶ Why Substructuring?
- ▶ Static Substructuring
- ▶ Advantages of Substructuring
- ▶ Procedures Supporting Substructures



30 minutes

# Lesson 2: Using Static Substructuring in Abaqus

## *Lesson content:*

- ▶ The Basics
- ▶ Substructure Generation
- ▶ Substructure Usage: Abaqus/CAE
- ▶ Substructure Usage: Keywords
- ▶ Substructure Load Cases
- ▶ Substructure Gravity Loading
- ▶ Kinematic Constraints in Substructures
- ▶ Flexible Body Dynamics
- ▶ Limitations



1 hour



# Lesson 3: Linear Perturbations about a Preloaded State

## *Lesson content:*

- ▶ Introduction
- ▶ Substructure Tangent Stiffness Calculation
- ▶ Response Quantities
- ▶ Effect of Preloads at the Usage Level
- ▶ Preloading Syntax
- ▶ Preloading Example: Rotating Structure



30 minutes

# Lesson 4: Dynamic Substructuring

## *Lesson content:*

- ▶ Guyan Reduction
- ▶ Dynamic Mode Addition
- ▶ Damping with Substructures



45 minutes

# Lesson 5: Substructure Output

## *Lesson content:*

- ▶ Introduction
- ▶ Visualizing Substructure Results
- ▶ Output of Eliminated Degrees of Freedom
- ▶ Output of Substructure Matrices
- ▶ Substructure Library Utilities



45 minutes

# Lesson 6: Substructuring Examples

## *Lesson content:*

- ▶ Cyclic Symmetry
- ▶ Multilevel Substructuring
- ▶ Workshop Preliminaries
- ▶ Workshop 1a: Substructures: Plane Frame Analysis (IA)
- ▶ Workshop 1a: Substructures: Plane Frame Analysis (KW)
- ▶ Workshop 1b: Substructures: Surface Mount Analysis (IA)
- ▶ Workshop 1b: Substructures: Surface Mount Analysis (KW)



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



3 hours

# Lesson 7: Using Substructures with Abaqus/Explicit

## *Lesson content:*

- ▶ Introduction
- ▶ Examples
- ▶ General Concepts
- ▶ Keyword Interface
- ▶ Interactive Interface
- ▶ Postprocessing
- ▶ Technology Notes
- ▶ Workshop 2: Substructures: Beam Impact (IA)
- ▶ Workshop 2: Substructures: Beam Impact (KW)



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



2 hours



# Lesson 8: Introduction to Submodeling

## *Lesson content:*

- ▶ Concept of Submodeling
- ▶ Motivation for Submodeling



15 minutes

# Lesson 9: Submodeling in Abaqus

## *Lesson content:*

- ▶ Fundamental Assumptions
- ▶ Submodeling Techniques
- ▶ Node-based Implementation
- ▶ Surface-based Implementation



30 minutes

# Lesson 10: Abaqus Usage and Examples (Part 1)

## *Lesson content:*

- ▶ Terminology
- ▶ Transfer of Data
- ▶ Prescribed Values
- ▶ Submodeling Workflow
- ▶ Surface-Based Submodel Boundaries
- ▶ Example: Conical Crack in a Half Space
- ▶ Example: Pressure Vessel
- ▶ Workshop 3: Submodeling: Pressure Vessel Nozzle Analysis (IA)
- ▶ Workshop 3: Submodeling: Pressure Vessel Nozzle Analysis (KW)



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



2.5 hours

# Lesson 11: Abaqus Usage and Examples (Part 2)

## *Lesson content:*

- ▶ Node-Based Submodel Boundaries
- ▶ Example: Stacked Sheet Metal Assembly
- ▶ Example: Large Displacement Analysis
- ▶ Tolerances at the Submodel Boundary
- ▶ Shell-to-Solid Submodeling
- ▶ Example: Shell-to-Solid Submodel of a Pipe Joint
- ▶ Workshop 4: Submodeling: Ceramic-Metal Braze Joint (IA)
- ▶ Workshop 4: Submodeling: Ceramic-Metal Braze Joint (KW)



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



2 hours

# Lesson 12: Submodeling Practices

## *Lesson content:*

- ▶ Perturbation Analysis
- ▶ Changing Procedures
- ▶ The Frequency Domain
- ▶ Submodeling and Thermal Stress Analysis
- ▶ Example: Thermal Strain in a Bar
- ▶ Submodeling in Dynamic Procedures
- ▶ Example: Speaker Diaphragm
- ▶ Workshop 5: Submodeling: Composite Tube Joint (IA)
- ▶ Workshop 5: Submodeling: Composite Tube Joint (KW)



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



1 hour



# Lesson 13: Limitations of Submodeling

## *Lesson content:*

- ▶ Elements
- ▶ Procedures
- ▶ Shell-to-Solid



15 minutes

# Appendix 1: Theory of Substructures

## *Appendix content:*

- ▶ Static Substructuring
- ▶ Guyan Reduction
- ▶ Restrained Mode Addition



15 minutes