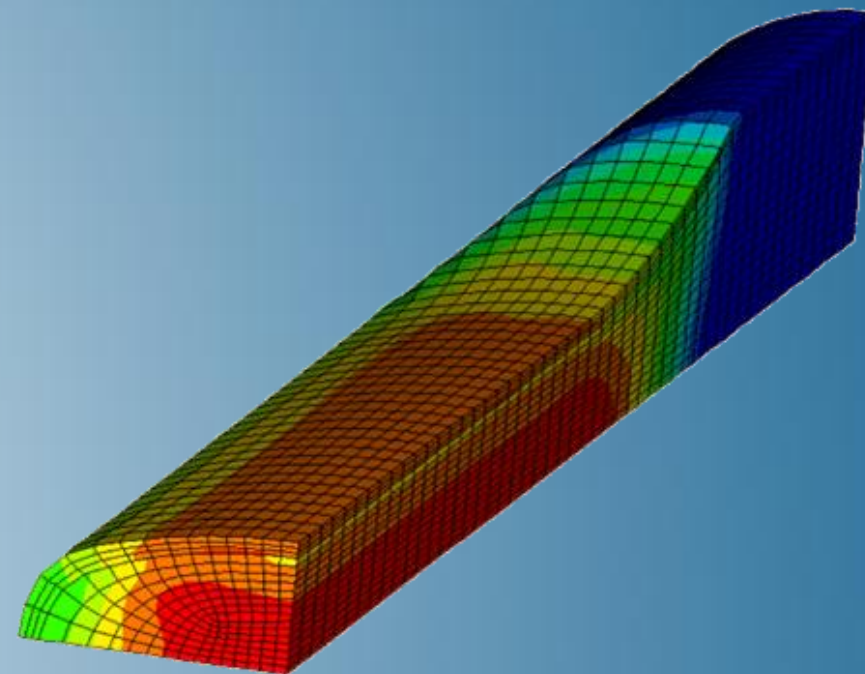


# Metal Forming with Abaqus

Abaqus 2018



**3DEXPERIENCE®**



# About this Course

## Course objectives

In this course you will learn practical modeling skills and techniques for:

- ▶ Stamping
- ▶ Hydroforming
- ▶ Punch stretching
- ▶ Forging
- ▶ Rolling
- ▶ Drawing
- ▶ Superplastic forming

## Targeted audience

This course is recommended for engineers with experience using Abaqus

## Prerequisites

None



3 days

# Day 1

---

- ▶ Lecture 1      Introduction
- ▶ Lecture 2      Solution Procedures in Abaqus
- ▶ Lecture 3      Contact
  - ▣ Workshop 1   Bulk Forming of a Cup
- ▶ Lecture 4      Elements
- ▶ Lecture 5      Materials

## Day 2

---

- ▶ Lecture 6            Adaptive Meshing
- ▶ Lecture 7            Modeling Quasi-Static Processes Using Abaqus/Explicit
  - Workshop 2   Rolling of a Thick Plate
- ▶ Lecture 8            Transferring Results between Abaqus Analyses
  - Workshop 3   Production of an Angle Bracket
- ▶ Lecture 9            Model Change
- ▶ Lecture 10           Thermal Effects

## Day 3 (Selected topics as time permits)

---

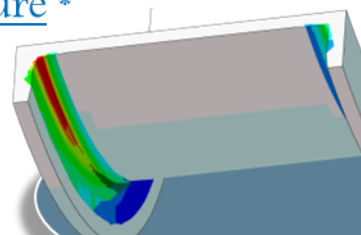
- ▶ Lecture 11      Rolling Analysis
- ▶ Lecture 12      Multi-Pass Rolling
- ▶ Lecture 13      Drawbead Modeling
- ▶ Lecture 14      Hydroforming
- ▶ Lecture 15      Superelastic Forming Analysis with Abaqus

# SIMULIA

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

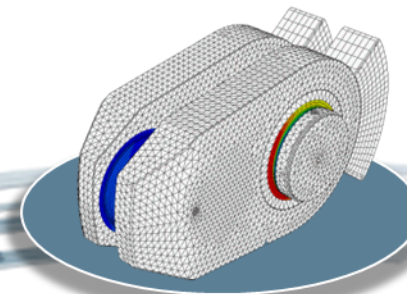
## Design Optimization: Tosca Structure \*

Simulation-driven design refinement to improve performance



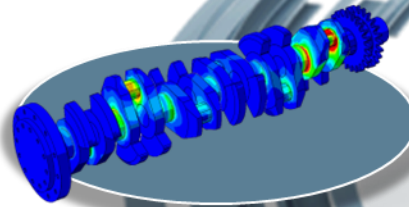
## Durability Assessment: fe-safe \*

Accurate life estimation to achieve certification



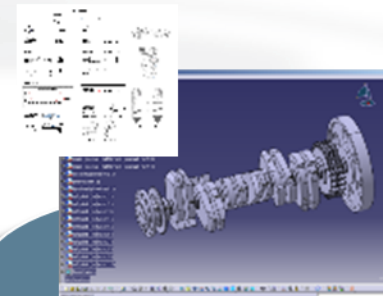
## FEA Stress Analysis: Abaqus \*

Detailed stress analysis using extracted load history from MBS



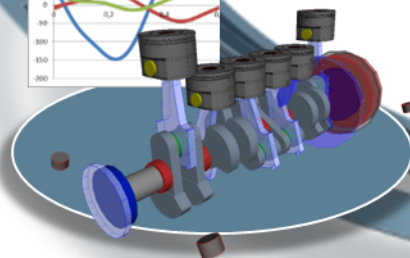
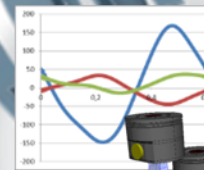
## CAD Geometry: CATIA

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



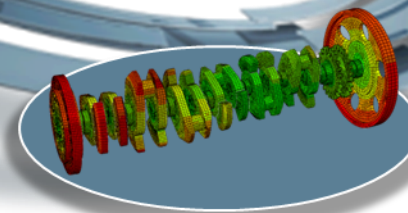
## Multibody Simulation: Simpack

System analysis to extract virtual load history of complete working cycle



## Mesh Calibration: Isight \*

Automated mesh calibration; sufficient mesh quality for accurate results

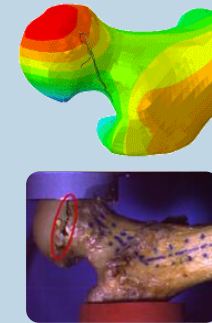


\* Included in extended licensing pool

# SIMULIA's Power of the Portfolio

## Abaqus

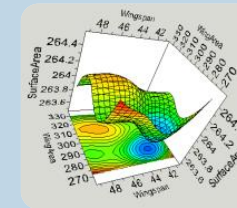
- Routine and Advanced Simulation
- Linear and Nonlinear, Static and Dynamic
- Thermal, Electrical, Acoustics
- Extended Physics through Co-simulation
- Model Preparation and Visualization



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

## Isight

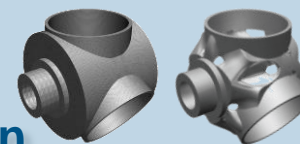
- Process Integration
- Design Optimization
- Parametric Optimization
- Six Sigma and Design of Experiments



**Material Calibration  
Workflow Automation  
Design Exploration**

## Tosca

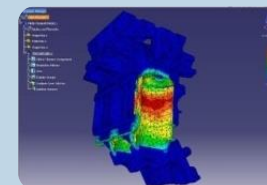
- Non-Parametric Optimization
- Structural and Fluid Flow Optimization
- Topology, Sizing, Shape, Bead Optimization



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

## fe-safe

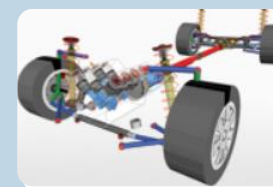
- Durability Simulation
- Low Cycle and High Cycle Fatigue
- Weld, High Temperature, Non-metallics



**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
Lecture 14	11/17	Updated for Abaqus 2018
Lecture 15	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

# Lesson 1: Introduction

## *Lesson content:*

- ▶ Introduction
- ▶ Forming Processes
- ▶ Motivation Behind Metal Forming Simulation



45 minutes

# Lesson 2: Solution Procedures with Abaqus

## *Lesson content:*

- ▶ Introduction
- ▶ Equilibrium
- ▶ Implicit Solution of Static Equilibrium
- ▶ Explicit Solution of Dynamic Equilibrium
- ▶ Implicit and Explicit Procedures for Metal Forming



45 minutes



# Lesson 3: Contact

## ***Lesson content:***

- ▶ Introduction to Modeling Contact
- ▶ Defining General Contact
- ▶ General Contact Output
- ▶ Limitations of General Contact
- ▶ Contact Pairs
- ▶ Contact Pair Output
- ▶ Contact Constraint Algorithm
- ▶ Friction
- ▶ Contact Modeling Tips
- ▶ Rigid Bodies in Abaqus
- ▶ Workshop Preliminaries
- ▶ Workshop 1: Bulk Forming of a Cup (IA)
- ▶ Workshop 1: Bulk Forming of a Cup (KW)



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



**3 hours**

# Lesson 4: Elements

## *Lesson content:*

- ▶ Introduction
- ▶ Continuum Elements
- ▶ Structural Elements
- ▶ Special-Purpose Elements
- ▶ Element Selection in Scoping Studies
- ▶ Hourglassing
- ▶ Secondary-order Accuracy



1.5 hours

# Lesson 5: Materials

## *Lesson content:*

- ▶ Introduction
- ▶ Elasticity
- ▶ Mises Plasticity
- ▶ Anisotropic (Hill's) Plasticity
- ▶ Gurson Model
- ▶ Rate Dependence
- ▶ Annealing
- ▶ Forming Limit Diagrams



1.5 hours

# Lesson 6: Adaptive Meshing

## *Lesson content:*

- ▶ Introduction to Adaptive Meshing
- ▶ Arbitrary Lagrangian-Eulerian (ALE) Method
- ▶ Lagrangian Adaptive Mesh Domains
- ▶ Eulerian Adaptive Mesh Domains for Steady-state Analyses
- ▶ Additional Features of Adaptive Meshing
- ▶ Adaptive Meshing Output and Diagnostics
- ▶ Summary



1.5 hours

# Lesson 7: Quasi-Static Analyses

## *Lesson content:*

- ▶ Introduction
- ▶ Quasi-Static Simulations Using Explicit Dynamics
- ▶ Loading Rates
- ▶ Energy Balance in Quasi-Static Analyses
- ▶ Mass Scaling
- ▶ Viscous Pressure
- ▶ Summary
- ▶ Workshop 2: Rolling of a Thick Plate (IA)
- ▶ Workshop 2: Rolling of a Thick Plate (KW)



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



2.5 hours



# Lesson 8: Transferring Results between Abaqus Analyses

## ***Lesson content:***

- ▶ Introduction
- ▶ Import from Abaqus/Explicit to Abaqus/Standard
- ▶ Import from Abaqus/Standard to Abaqus/Explicit
- ▶ Import from Abaqus/Explicit to Abaqus/Explicit
- ▶ Additional Import Modeling Issues
- ▶ Limitations
- ▶ Demonstration
- ▶ Workshop 3: Production of an Angle Bracket (IA)
- ▶ Workshop 3: Production of an Angle Bracket (KW)



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



**2 hours**

# Lesson 9: Model Change

## *Lesson content:*

- ▶ Introduction
- ▶ Multistage Forming Processes
- ▶ Element Removal in Abaqus/Standard



30 minutes

# Lesson 10: Thermal Effects

## *Lesson content:*

- ▶ Introduction
- ▶ Fully Coupled Analysis
- ▶ Adiabatic Analysis
- ▶ Rigid Bodies in Thermal-Stress Analysis



45 minutes

# Lesson 11: Rolling Analysis

## *Lesson content:*

- ▶ Introduction
- ▶ Hot Rolling with Abaqus/Explicit
- ▶ Single Pass Simulation
- ▶ Steady-State Analysis
- ▶ Transient Analysis



45 minutes

# Lesson 12: Multi-Pass Rolling Analysis

## *Lesson content:*

- ▶ Multi-Pass Simulation



30 minutes



# Lesson 13: Drawbead Modeling

## *Lesson content:*

- ▶ Introduction
- ▶ Drawbead Restraint Forces
- ▶ Nonlinear Springs
- ▶ Point Masses with Node-Based Contact
- ▶ Example: Forming of a Fender



45 minutes

# Lesson 14: Hydroforming

## *Lesson content:*

- ▶ Introduction
- ▶ Hydroforming Processes
- ▶ Hydroforming Example



30 minutes

# Lesson 15: Superplastic Forming Analysis with Abaqus

## *Lesson content:*

- ▶ Introduction
- ▶ Superplastic Forming Modeling
- ▶ Superplastic Forming Example



30 minutes