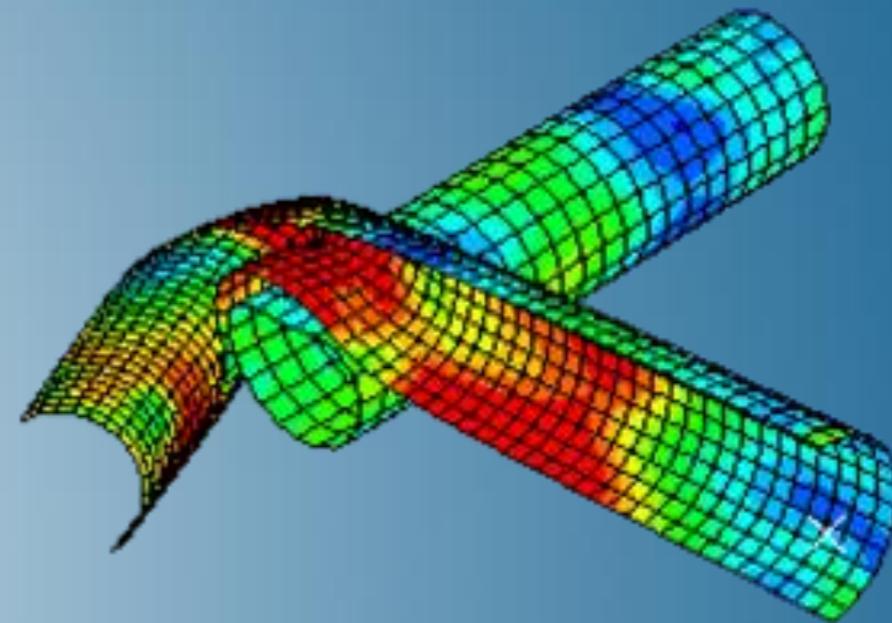


Introduction to Abaqus/Standard and Abaqus/Explicit

Abaqus 2018



3DEXPERIENCE[®]



About this Course

Course objectives

Upon completion of this course you will be able to:

- ▶ Complete finite element models using Abaqus keywords.
- ▶ Submit and monitor analysis jobs.
- ▶ View and evaluate simulation results.
- ▶ Solve structural analysis problems using Abaqus/Standard and Abaqus/Explicit, including the effects of material nonlinearity, large deformation and contact.

Targeted audience

Simulation Analysts

Prerequisites

None



3 days

Day 1

Lesson 1	Defining an Abaqus Model
Workshop 1	Basic Input and Output
Lesson 2	Linear Static Analysis
Workshop 2	Linear Static Analysis of a Cantilever Beam: Multiple Load Cases
Lesson 3	Nonlinear Analysis in Abaqus/Standard
Workshop 3	Nonlinear Statics

Day 2

Lesson 4 Multistep Analysis in Abaqus

Workshop 4 Unloading Analysis

Lesson 5 Constraints and Contact

Workshop 5 Seal Contact

Lesson 6 Introduction to Dynamics

Workshop 6 Dynamics

Day 3

Lesson 7	Using Abaqus/Explicit
Workshop 7	Contact with Abaqus/Explicit
Lesson 8	Quasi-Static Analysis in Abaqus/Explicit
Workshop 8	Quasi-Static Analysis (<i>Optional</i>)
Lesson 9	Combining Abaqus/Standard & Abaqus/Explicit
Workshop 9	Import Analysis (<i>Optional</i>)

Additional Material

Appendix 1

Element Selection Criteria

Appendix 2

Contact Issues Specific to Abaqus/Standard

Appendix 3

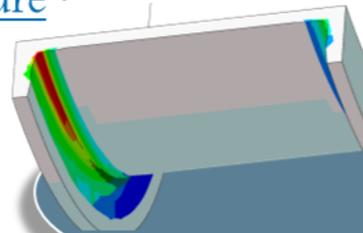
Contact Issues Specific to Abaqus/Explicit

SIMULIA

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

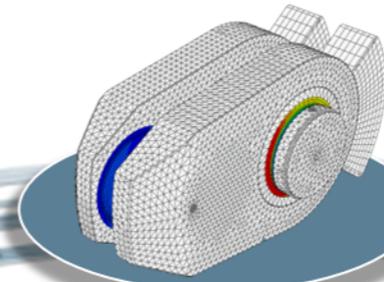
Design Optimization: Tosca Structure *

Simulation-driven design refinement to improve performance



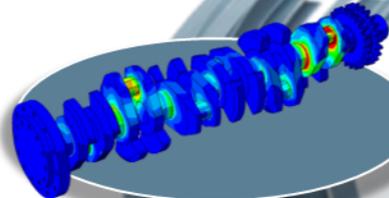
Durability Assessment: fe-safe *

Accurate life estimation to achieve certification



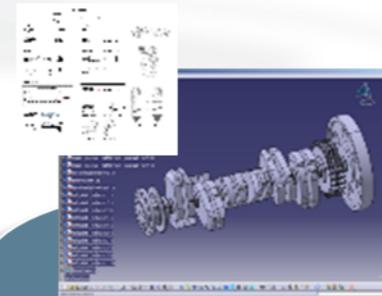
FEA Stress Analysis: Abaqus *

Detailed stress analysis using extracted load history from MBS



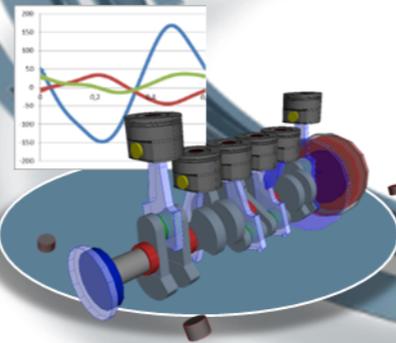
CAD Geometry: CATIA

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



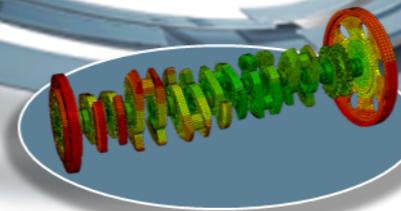
Multibody Simulation: Simpack

System analysis to extract virtual load history of complete working cycle



Mesh Calibration: Isight *

Automated mesh calibration; sufficient mesh quality for accurate results

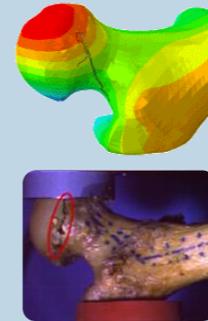


* Included in extended licensing pool

SIMULIA's Power of the Portfolio

Abaqus

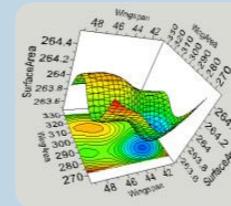
- Routine and Advanced Simulation
- Linear and Nonlinear, Static and Dynamic
- Thermal, Electrical, Acoustics
- Extended Physics through Co-simulation
- Model Preparation and Visualization



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

Isight

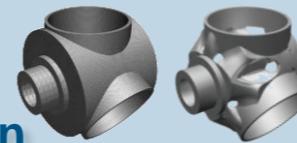
- Process Integration
- Design Optimization
- Parametric Optimization
- Six Sigma and Design of Experiments



**Material Calibration
Workflow Automation
Design Exploration**

Tosca

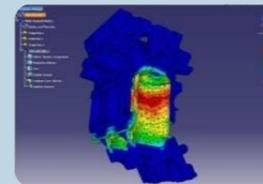
- Non-Parametric Optimization
- Structural and Fluid Flow Optimization
- Topology, Sizing, Shape, Bead Optimization



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

fe-safe

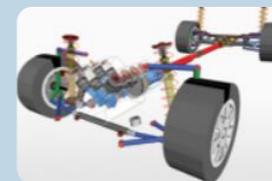
- Durability Simulation
- Low Cycle and High Cycle Fatigue
- Weld, High Temperature, Non-metallics



**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.

 | The 3DEXPERIENCE Company

SIMULIA Training

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

Home ... SIMULIA SERVICES TRAINING COURSES SCHEDULE & REGISTRATION

SIMULIA

in f t YouTube

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

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

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
Workshop 9	11/17	Updated for Abaqus 2018

Lesson 1: Defining an Abaqus Model

Lesson content:

- ▶ Introduction
- ▶ Abaqus FEA
- ▶ Abaqus/CAE
- ▶ Abaqus/Standard and Abaqus/Explicit
- ▶ Documentation
- ▶ SIMULIA Learning Community
- ▶ Components of an Abaqus Model
- ▶ Details of an Abaqus Input File
- ▶ Abaqus Conventions
- ▶ Output
- ▶ Example: Cantilever Beam Model
- ▶ Parts and Assemblies (optional)
- ▶ Workshop Preliminaries
- ▶ Workshop 1: Basic Input and Output (IA)
- ▶ Workshop 1: Basic Input and Output (KW)



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



2 hours

Lesson 2: Linear Static Analysis

Lesson content:

- ▶ Linear and Nonlinear Procedures
- ▶ Linear Static Analysis and Multiple Load Cases
- ▶ Multiple Load Case Usage
- ▶ Examples
- ▶ Workshop 2: Linear Static Analysis of a Cantilever Beam (IA)
- ▶ Workshop 2: Linear Static Analysis of a Cantilever Beam (KW)



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



2 hours

Lesson 3: Nonlinear Analysis in Abaqus

Lesson content:

- ▶ Nonlinearity in Structural Mechanics
- ▶ Equations of Motion
- ▶ Nonlinear Analysis Using Implicit Methods
- ▶ Nonlinear Analysis Using Explicit Methods
- ▶ Input File for Nonlinear Analysis
- ▶ Status File
- ▶ Message File
- ▶ Output from Nonlinear Cantilever Beam Analysis
- ▶ Workshop 3: Nonlinear Statics (IA)
- ▶ Workshop 3: Nonlinear Statics (KW)



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



2 hours

Lesson 4: Multistep Analysis in Abaqus

Lesson content:

- ▶ Multistep Analyses
- ▶ Restart Analysis in Abaqus
- ▶ Workshop 4: Unloading Analysis (IA)
- ▶ Workshop 4: Unloading Analysis (KW)



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



1 hour

Lesson 5: Constraints and Contact

Lesson content:

- ▶ Constraints
- ▶ Tie Constraints
- ▶ Rigid Bodies
- ▶ Shell-to-solid Coupling
- ▶ Contact
- ▶ Defining General Contact
- ▶ Defining Contact Pairs
- ▶ Contact Pair Surfaces
- ▶ Local Surface Behavior
- ▶ Relative Sliding of Points in Contact
- ▶ Adjusting Initial Nodal Locations for Contact
- ▶ Contact Output
- ▶ Workshop 5: Seal Contact (IA)
- ▶ Workshop 5: Seal Contact (KW)



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



2.5 hours

Lesson 6: Introduction to Dynamics

Lesson content:

- ▶ What Makes a Problem Dynamic?
- ▶ Equations for Dynamic Problems
- ▶ Linear Dynamics
- ▶ Nonlinear Dynamics
- ▶ Comparing Abaqus/Standard and Abaqus/Explicit
- ▶ Nonlinear Dynamics Example
- ▶ Workshop 6: Dynamics (IA)
- ▶ Workshop 6: Dynamics (KW)



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



2 hours

Lesson 7: Using Abaqus/Explicit

Lesson content:

- ▶ Overview of the Explicit Dynamics Procedure
- ▶ Abaqus/Explicit Syntax
- ▶ Rigid Bodies
- ▶ Workshop 7: Contact with Abaqus/Explicit (IA)
- ▶ Workshop 7: Contact with Abaqus/Explicit (KW)



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



2 hours

Lesson 8: Quasi-Static Analysis in Abaqus/Explicit

Lesson content:

- ▶ Introduction
- ▶ Solution Strategies
- ▶ Quasi-Static Simulations Using Explicit Dynamics
- ▶ Energy Balance
- ▶ Example: Load Rates
- ▶ Example: Mass Scaling
- ▶ Adaptive Meshing
- ▶ Workshop 8: Quasi-Static Analysis (IA)
- ▶ Workshop 8: Quasi-Static Analysis (KW)



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



2 hours

Lesson 9: Combining Abaqus/Standard & Abaqus/Explicit

Lesson content:

- ▶ Introduction
- ▶ Abaqus Usage
- ▶ Springback Calculation using Abaqus/Standard
- ▶ Workshop 9: Import Analysis (IA)
- ▶ Workshop 9: Import Analysis (KW)



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



1 hour

Appendix 1: Element Selection Criteria

Appendix content:

- ▶ Elements
- ▶ Structural (Shells and Beams) vs. Continuum Elements
- ▶ Modeling Bending Using Continuum Elements
- ▶ Stress Concentrations
- ▶ Contact
- ▶ Incompressible Materials
- ▶ Mesh Generation
- ▶ Solid Element Selection Summary



1.5 hours

Appendix 2: Contact Issues Specific to Abaqus/Standard

Appendix content:

- ▶ Contact as Part of the Model Definition
- ▶ Mesh Density Considerations
- ▶ Contact Logic in Abaqus/Standard



30 minutes

Appendix 3: Contact Issues Specific to Abaqus/Explicit

Appendix content:

- ▶ Contact Pairs as Part of the History Data
- ▶ Enforcing the Contact Constraints
- ▶ Double-Sided Contact
- ▶ Initial Kinematic Compliance



30 minutes