Metal Forming with Abaqus

3DEXPERIENCE
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
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 is the Dassault Systèmes brand for Realistic Simulation solutions

Portfolio of established, best-in-class products

- Abaqus, Isight, Tosca, fe-safe

- All using a common extended licensing pool
SIMULIA’s Power of the Portfolio

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

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

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

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

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

**Material Calibration**
- Workflow Automation
- Design Exploration

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

**Safety Factors**
- Creep-Fatigue Interaction
- Weld Fatigue
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!

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

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, 2015

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 Abaqus Installation and Licensing Guide.
<table>
<thead>
<tr>
<th>Lecture</th>
<th>Date</th>
<th>Update Status</th>
</tr>
</thead>
<tbody>
<tr>
<td>Lecture 1</td>
<td>11/15</td>
<td>Updated for Abaqus 2016</td>
</tr>
<tr>
<td>Lecture 2</td>
<td>11/15</td>
<td>Updated for Abaqus 2016</td>
</tr>
<tr>
<td>Lecture 3</td>
<td>11/15</td>
<td>Updated for Abaqus 2016</td>
</tr>
<tr>
<td>Lecture 4</td>
<td>11/15</td>
<td>Updated for Abaqus 2016</td>
</tr>
<tr>
<td>Lecture 5</td>
<td>11/15</td>
<td>Updated for Abaqus 2016</td>
</tr>
<tr>
<td>Lecture 6</td>
<td>11/15</td>
<td>Updated for Abaqus 2016</td>
</tr>
<tr>
<td>Lecture 7</td>
<td>11/15</td>
<td>Updated for Abaqus 2016</td>
</tr>
<tr>
<td>Lecture 8</td>
<td>11/15</td>
<td>Updated for Abaqus 2016</td>
</tr>
<tr>
<td>Lecture 9</td>
<td>11/15</td>
<td>Updated for Abaqus 2016</td>
</tr>
<tr>
<td>Lecture 10</td>
<td>11/15</td>
<td>Updated for Abaqus 2016</td>
</tr>
<tr>
<td>Lecture 11</td>
<td>11/15</td>
<td>Updated for Abaqus 2016</td>
</tr>
<tr>
<td>Lecture 12</td>
<td>11/15</td>
<td>Updated for Abaqus 2016</td>
</tr>
<tr>
<td>Lecture 13</td>
<td>11/15</td>
<td>Updated for Abaqus 2016</td>
</tr>
<tr>
<td>Lecture 14</td>
<td>11/15</td>
<td>Updated for Abaqus 2016</td>
</tr>
<tr>
<td>Lecture 15</td>
<td>11/15</td>
<td>Updated for Abaqus 2016</td>
</tr>
<tr>
<td>Workshop 1</td>
<td>11/15</td>
<td>Updated for Abaqus 2016</td>
</tr>
<tr>
<td>Workshop 2</td>
<td>11/15</td>
<td>Updated for Abaqus 2016</td>
</tr>
<tr>
<td>Workshop 3</td>
<td>11/15</td>
<td>Updated for Abaqus 2016</td>
</tr>
</tbody>
</table>
Lesson 1: Introduction

Lesson content:

- Introduction
- Forming Processes
- Motivation Behind Metal Forming Simulation
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.
Lesson content:

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

Lesson 4: Elements
1.5 hours
Lesson content:

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

Lesson 5: Materials

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
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.
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
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
Lesson 13: Drawbead Modeling

Lesson content:

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