Metal Forming with Abaqus

6.14
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
Join the Community!

How can you maximize the robust technology of Abaqus FEA and Isight?
Connect with peers to share knowledge and get technical insights

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.

©2013 Dassault Systemes. All rights reserved.
Legal Notices

The Abaqus Software described in this documentation is available only under license from Dassault Systèmes or its subsidiary 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 subsidiary.

© Dassault Systèmes, 2014

Printed in the United States of America.

Abaqus, the 3DS logo, SIMULIA, and CATIA 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 6.14 Installation and Licensing Guide.
<p>| | | |</p>
<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Lecture 1</td>
<td>5/14</td>
<td>Updated for 6.14</td>
</tr>
<tr>
<td>Lecture 2</td>
<td>5/14</td>
<td>Updated for 6.14</td>
</tr>
<tr>
<td>Lecture 3</td>
<td>5/14</td>
<td>Updated for 6.14</td>
</tr>
<tr>
<td>Lecture 4</td>
<td>5/14</td>
<td>Updated for 6.14</td>
</tr>
<tr>
<td>Lecture 5</td>
<td>5/14</td>
<td>Updated for 6.14</td>
</tr>
<tr>
<td>Lecture 6</td>
<td>5/14</td>
<td>Updated for 6.14</td>
</tr>
<tr>
<td>Lecture 7</td>
<td>5/14</td>
<td>Updated for 6.14</td>
</tr>
<tr>
<td>Lecture 8</td>
<td>5/14</td>
<td>Updated for 6.14</td>
</tr>
<tr>
<td>Lecture 9</td>
<td>5/14</td>
<td>Updated for 6.14</td>
</tr>
<tr>
<td>Lecture 10</td>
<td>5/14</td>
<td>Updated for 6.14</td>
</tr>
<tr>
<td>Lecture 11</td>
<td>5/14</td>
<td>Updated for 6.14</td>
</tr>
<tr>
<td>Lecture 12</td>
<td>5/14</td>
<td>Updated for 6.14</td>
</tr>
<tr>
<td>Lecture 13</td>
<td>5/14</td>
<td>Updated for 6.14</td>
</tr>
<tr>
<td>Lecture 14</td>
<td>5/14</td>
<td>Updated for 6.14</td>
</tr>
<tr>
<td>Lecture 15</td>
<td>5/14</td>
<td>Updated for 6.14</td>
</tr>
<tr>
<td>Workshop 1</td>
<td>5/14</td>
<td>Updated for 6.14</td>
</tr>
<tr>
<td>Workshop 2</td>
<td>5/14</td>
<td>Updated for 6.14</td>
</tr>
<tr>
<td>Workshop 3</td>
<td>5/14</td>
<td>Updated for 6.14</td>
</tr>
</tbody>
</table>
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
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 4: Elements

Lesson content:

- Introduction
- Continuum Elements
- Structural Elements
- Special-Purpose Elements
- Element Selection in Scoping Studies
- Rigid Bodies
- 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 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
- Additional Import Modeling Issues
- Limitations
- Import from Abaqus/Explicit to Abaqus/Explicit
- 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.
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 content:

- Multi-Pass Simulation
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 content:

- Introduction
- Superplastic Forming Modeling
- Superplastic Forming Example