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, Simpack

* 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

### 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

### Simpack
- 3D Multibody Dynamics Simulation
- Mechanical or Mechatronic Systems
- Detailed Transient Simulation (Offline and Realtime)

### 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

### 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!

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 Systèmes. All rights reserved.
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
## Revision Status

<table>
<thead>
<tr>
<th>Lecture</th>
<th>Date</th>
<th>Update Details</th>
</tr>
</thead>
<tbody>
<tr>
<td>Lecture 1</td>
<td>11/16</td>
<td>Updated for Abaqus 2017</td>
</tr>
<tr>
<td>Lecture 2</td>
<td>11/16</td>
<td>Updated for Abaqus 2017</td>
</tr>
<tr>
<td>Lecture 3</td>
<td>11/16</td>
<td>Updated for Abaqus 2017</td>
</tr>
<tr>
<td>Lecture 4</td>
<td>11/16</td>
<td>Updated for Abaqus 2017</td>
</tr>
<tr>
<td>Lecture 5</td>
<td>11/16</td>
<td>Updated for Abaqus 2017</td>
</tr>
<tr>
<td>Lecture 6</td>
<td>11/16</td>
<td>Updated for Abaqus 2017</td>
</tr>
<tr>
<td>Lecture 7</td>
<td>11/16</td>
<td>Updated for Abaqus 2017</td>
</tr>
<tr>
<td>Lecture 8</td>
<td>11/16</td>
<td>Updated for Abaqus 2017</td>
</tr>
<tr>
<td>Lecture 9</td>
<td>11/16</td>
<td>Updated for Abaqus 2017</td>
</tr>
<tr>
<td>Lecture 10</td>
<td>11/16</td>
<td>Updated for Abaqus 2017</td>
</tr>
<tr>
<td>Lecture 11</td>
<td>11/16</td>
<td>Updated for Abaqus 2017</td>
</tr>
<tr>
<td>Lecture 12</td>
<td>11/16</td>
<td>Updated for Abaqus 2017</td>
</tr>
<tr>
<td>Lecture 13</td>
<td>11/16</td>
<td>Updated for Abaqus 2017</td>
</tr>
<tr>
<td>Lecture 14</td>
<td>11/16</td>
<td>Updated for Abaqus 2017</td>
</tr>
<tr>
<td>Lecture 15</td>
<td>11/16</td>
<td>Updated for Abaqus 2017</td>
</tr>
<tr>
<td>Workshop 1</td>
<td>11/16</td>
<td>Updated for Abaqus 2017</td>
</tr>
<tr>
<td>Workshop 2</td>
<td>11/16</td>
<td>Updated for Abaqus 2017</td>
</tr>
<tr>
<td>Workshop 3</td>
<td>11/16</td>
<td>Updated for Abaqus 2017</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 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
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.

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.
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 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
Lesson 15: Superplastic Forming Analysis with Abaqus

Lesson content:

- Introduction
- Superplastic Forming Modeling
- Superplastic Forming Example

30 minutes