Course objectives
In this course you will learn about:

- When and how to use subroutines
- DLOAD, VDLOAD, and UTRACLOAD for specifying user-defined loading
- FILM for specifying user-defined film conditions
- USDFLD and VUSDFLD for defining field variable dependence
- UVARM for defining a user output variable
- UHYPER for modeling hyperelastic materials
- UMAT and VUMAT for allowing constitutive models to be added to the program
- UEL and VUEL for allowing the creation of user-defined elements

Targeted audience
This course is recommended for engineers with experience using Abaqus.

Prerequisites
A working knowledge of the finite element method and programming in either Fortran or C
Day 1

- Lecture 1  Introduction
- Lecture 2  User Subroutines (V)DLOAD and UTRACLOAD
- Lecture 3  User Subroutine FILM
- Workshop 1  User Subroutine FILM
- Lecture 4  User Subroutine (V)USDFLD
- Lecture 5  User Subroutine UVARM
Day 2

- Lecture 6  User Subroutine UHYPER
- Lecture 7  Writing a UMAT or VUMAT
- Workshop 2  User Subroutine UMAT: Tangent Stiffness
- Lecture 8  Creating a Nonlinear User Element (UEL and VUEL)
Additional Material

- Appendix 1  Logical Modeling in Abaqus
- Workshop 3  Controlling an Inverted Pendulum with VUAMP
- Appendix 2  User Subroutine URDFIL
- Appendix 3  User Subroutine (V)UANISOHYPER
- Workshop 4  User Subroutine UANISOHYPER_INV: anisotropic hyperelastic material behavior
- Appendix 4  Introduction to Parallel Computing
- Appendix 5  Getting Started with Abaqus Parallel Execution
  - Workshop 5  User Subroutines with Threads
  - Workshop 6  User Subroutines with MPI
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

<table>
<thead>
<tr>
<th><strong>Abaqus</strong></th>
<th><strong>Isight</strong></th>
<th><strong>Tosca</strong></th>
<th><strong>fe-safe</strong></th>
<th><strong>Simpack</strong></th>
</tr>
</thead>
</table>
| • Routine and Advanced Simulation  
• Linear and Nonlinear, Static and Dynamic  
• Thermal, Electrical, Acoustics  
• Extended Physics through Co-simulation  
• Model Preparation and Visualization | • Process Integration  
• Design Optimization  
• Parametric Optimization  
• Six Sigma and Design of Experiments | • Non-Parametric Optimization  
• Structural and Fluid Flow Optimization  
• Topology, Sizing, Shape, Bead Optimization | • Durability Simulation  
• Low Cycle and High Cycle Fatigue  
• Weld, High Temperature, Non-metallics | • 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!
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.
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, 2016

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

<table>
<thead>
<tr>
<th>Component</th>
<th>Date</th>
<th>Status</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>Appendix 1</td>
<td>11/16</td>
<td>Updated for Abaqus 2017</td>
</tr>
<tr>
<td>Appendix 2</td>
<td>11/16</td>
<td>Updated for Abaqus 2017</td>
</tr>
<tr>
<td>Appendix 3</td>
<td>11/16</td>
<td>Updated for Abaqus 2017</td>
</tr>
<tr>
<td>Appendix 4</td>
<td>11/16</td>
<td>New for Abaqus 2017</td>
</tr>
<tr>
<td>Appendix 5</td>
<td>11/16</td>
<td>New 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>
<tr>
<td>Workshop 4</td>
<td>11/16</td>
<td>Updated for Abaqus 2017</td>
</tr>
<tr>
<td>Workshop 5</td>
<td>11/16</td>
<td>New for Abaqus 2017</td>
</tr>
<tr>
<td>Workshop 6</td>
<td>11/16</td>
<td>New for Abaqus 2017</td>
</tr>
</tbody>
</table>
Lesson content:

- Overview of Some User Subroutines
- Where User Subroutines Fit into Abaqus/Standard
- Including User Subroutines in a Model
- Writing Output from User Subroutines
- Compiling and Linking User Subroutines
- Debugging Techniques and Proper Programming Habits
- C/C++ interface
- Property and Parameter Tables
- Support for User Subroutines
Lesson 2: User Subroutines (V)DLOAD and UTRACLOAD

Lesson content:

- Introduction
- Abaqus Usage
- DLOAD Subroutine Interface
- Example: Viscoelastic Cylinder
- Example: Asymmetric Pressure Loads
- VDLOAD Subroutine Interface
- Example: Viscoelastic Cylinder Revisited
- UTRACLOAD Subroutine Interface
- Example: Flexure of a Cantilever Beam
Lesson 3: User Subroutine FILM

Lesson content:

- Introduction
- Abaqus Usage
- FILM Subroutine Interface
- Example: Radiation in Finned Surface
- Workshop Preliminaries
- Workshop 1: User Subroutine FILM (IA)
- Workshop 1: User Subroutine FILM (KW)

Both interactive (IA) and keywords (KW) versions of the workshop are provided. Complete only one.
Lesson 4: User Subroutine (V)USDFLD

Lesson content:

- Introduction
- Abaqus Usage
- Utility Routine GETVRM
- USDFLD Subroutine Interface
- Example: Laminated Composite Plate Failure
- Utility Routine VGETVRM
- VUSDFLD Subroutine Interface
- Example: Laminated Composite Plate Failure Revisited

2 hours
Lesson content:

- Introduction
- Abaqus Usage
- UVARM Subroutine Interface
- Example 1: Calculation of Stress Relative to Shift Tensor
- Example 2: Creating Contour Plots for UELs
Lesson 6: User Subroutine UHYPER

Lesson content:

- Overview
- Motivation
- Steps Required in Writing a UHYPER Interface
- Example 1: UHYPER for Neo-Hookean Hyperelasticity
Lesson 8: Creating a Nonlinear User Element

Lesson content:

- Overview
- Motivation
- Defining a User Element in Abaqus/Standard
- UEL Interface
- Example 1: Planar Beam Element with Nonlinear Section Behavior
- Example 2: Force Control Element
- Example 3: Plane Strain Element
- UELMAT Interface
- Using Nonlinear User Elements in Various Analysis Procedures
- Defining a User Element in Abaqus/Explicit
- VUEL Interface
- Example 4: Three-Dimensional Truss Element

3 hours
Appendix 1: Logical Modeling in Abaqus

Appendix content:

- Introduction
- Defining Logical Modeling
- Example: Force Control
- Workshop 3: Controlling an Inverted Pendulum with VUAMP (IA)
- Workshop 3: Controlling an Inverted Pendulum with VUAMP (KW)

Both interactive (IA) and keywords (KW) versions of the workshop are provided. Complete only one.
Appendix 2: User Subroutine URDFIL

Appendix content:

- Introduction
- Abaqus Usage
- URDFIL Subroutine Interface
- Example: Using URDFIL to Terminate an Analysis
Appendix 3: User Subroutine $(V)UANISOHYPER$

Appendix content:

- Overview
- Motivation
- Steps Required in Writing a UANISIOHYPER or VUANISOHYPER
- UANISOHYPER\_INV interface
- Example 1: UANISOHYPER\_INV for Kaliske and Schmidt
- UANISOHYPER\_STRAIN Interface
- Example 2: UANISOHYPER\_STRAIN for Saint-Venant Kirchhoff
- VUANISOHYPER\_INV interface
- Example 3: VUANISOHYPER\_INV for Kaliske and Schmidt
- VUANISOHYPER\_STRAIN Interface
- Example 4: VUANISOHYPER\_STRAIN for Saint-Venant Kirchhoff
- Workshop 4: User Subroutine UANISOHYPER\_INV: Anisotropic hyperelastic material behavior

2 hours
Appendix 4: Introduction to Parallel Computing

Appendix content:

- Overview
- What is Parallel Computing?
- Why do we use Parallel Computing?
- Computer Memory Architecture Basics
- Different Models of Parallel Computing
- Limits of Parallel Computing
- Challenges of Parallel Computing
- Abaqus Performance Benchmark

1 hour
Appendix 5: Getting Started with Abaqus Parallel Execution

Appendix content:

- Overview
- Parallel Execution in User Subroutines
- User Subroutines with Threads
- User Subroutines with MPI
- User Subroutines with hybrid MPI and Threads
- Workshop 5: User Subroutines with Threads
- Workshop 6: User Subroutines with MPI

2.5 hours