Element Selection in Abaqus

Abaqus 2018
Course objectives
Upon completion of this course you will be able to:

- Understand the distinguishing characteristics of the wide range of continuum and structural elements available in Abaqus for stress analyses
- Understand modeling features that may cause certain types of elements to behave poorly
- Choose appropriate element types for different applications including the effects of fully or nearly incompressible material behavior, contact, bending, etc.

Targeted audience
Simulation Analysts

Prerequisites
This course is recommended for engineers with experience using Abaqus

2 days
Day 1

- **Lecture 1** Solid Elements in Abaqus
- **Lecture 2** Other Solid Element Types (*optional*)
- **Lecture 3** Integration, Hourglassing, & Incompressibility
  - **Demo 1** Hourglass control
  - **Workshop 1** Plane Strain Elements
- **Lecture 4** Key Properties of Solid Elements
  - **Workshop 2** Plane Stress Elements
- **Lecture 5** Modeling Bending and Stress Concentrations
  - **Workshop 3** 3-D Solid Elements
Day 2

- Lecture 6  Structural Elements in Abaqus
- Lecture 7  Conventional Shell Elements
  - Workshop 4  Shell Elements
- Lecture 8  Continuum Shell Elements
- Lecture 9  Beam and Frame Elements
  - Workshop 5  Beam Elements
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
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.

©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
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.
<table>
<thead>
<tr>
<th>Revision Status</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
</tr>
<tr>
<td>Lecture 1</td>
</tr>
<tr>
<td>11/17 Updated for Abaqus 2018</td>
</tr>
<tr>
<td>Lecture 2</td>
</tr>
<tr>
<td>11/17 Updated for Abaqus 2018</td>
</tr>
<tr>
<td>Lecture 3</td>
</tr>
<tr>
<td>11/17 Updated for Abaqus 2018</td>
</tr>
<tr>
<td>Lecture 4</td>
</tr>
<tr>
<td>11/17 Updated for Abaqus 2018</td>
</tr>
<tr>
<td>Lecture 5</td>
</tr>
<tr>
<td>11/17 Updated for Abaqus 2018</td>
</tr>
<tr>
<td>Lecture 6</td>
</tr>
<tr>
<td>11/17 Updated for Abaqus 2018</td>
</tr>
<tr>
<td>Lecture 7</td>
</tr>
<tr>
<td>11/17 Updated for Abaqus 2018</td>
</tr>
<tr>
<td>Lecture 8</td>
</tr>
<tr>
<td>11/17 Updated for Abaqus 2018</td>
</tr>
<tr>
<td>Lecture 9</td>
</tr>
<tr>
<td>11/17 Updated for Abaqus 2018</td>
</tr>
<tr>
<td>Demo 1</td>
</tr>
<tr>
<td>11/17 Updated for Abaqus 2018</td>
</tr>
<tr>
<td>Workshop 1</td>
</tr>
<tr>
<td>11/17 Updated for Abaqus 2018</td>
</tr>
<tr>
<td>Workshop 2</td>
</tr>
<tr>
<td>11/17 Updated for Abaqus 2018</td>
</tr>
<tr>
<td>Workshop 3</td>
</tr>
<tr>
<td>11/17 Updated for Abaqus 2018</td>
</tr>
<tr>
<td>Workshop 4</td>
</tr>
<tr>
<td>11/17 Updated for Abaqus 2018</td>
</tr>
<tr>
<td>Workshop 5</td>
</tr>
<tr>
<td>11/17 Updated for Abaqus 2018</td>
</tr>
</tbody>
</table>
Lesson 1: Solid Elements in Abaqus

Lesson content:

- Introduction
- Solid Elements in Abaqus
- Planar Elements
- Generalized Plane Strain Elements (S)
- Axisymmetric Elements
- Axisymmetric Elements with Nonaxisymmetric Response (S)
- Axisymmetric Elements with Twist (S)
- Three-dimensional Elements

(S) Abaqus/Standard only

1.5 hours
Lesson 2: Other Solid Element Types

Lesson content:

- Cylindrical Elements (S)
- Infinite Elements
- Continuum Solid Shell Elements (S)

This lecture is optional

(S) Abaqus/Standard only

1 hour
Lesson content:

- Numerical Integration
- Hourglassing and Hourglass Control
- Incompressible Material Behavior
- The $\mathbf{B}$ Approach
- Hybrid (Mixed) Elements (S)
- Workshop Preliminaries
- Demonstration 1: Hourglass control
- Workshop 1: Plane Strain Elements (IA)
- Workshop 1: Plane Strain Elements (KW)

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

(S) Abaqus/Standard only
Lesson content:

- First-Order Triangles, Tetrahedra, Pyramids and Wedges
- Fully Integrated First-Order Quads and Bricks
- Reduced-Integration First-Order Quads and Bricks
- Second-Order Triangles and Tetrahedra
- Fully Integrated Second-Order Quads and Bricks (S)
- Reduced-Integration Second-Order Quads and Bricks (S)
- Using Second-Order Bricks in Contact Problems (S)
- Quads/Bricks or Triangles/Tetrahedra?
- First- or Second-Order Elements?
- Workshop 2: Plane Stress Elements (IA)
- Workshop 2: Plane Stress Elements (KW)

Both interactive (IA) and keywords (KW) versions of the workshop are provided. Complete only one.
Lesson 5: Modeling Bending and Stress Concentrations

Lesson content:

- Bending Problems and Shear Locking
- Incompatible Mode Elements
- Summary: Modeling Bending Using Continuum Elements
- Stress Concentrations
- Second-order Accuracy (E)
- Solid Element Selection Summary
- Abaqus/Standard vs. Abaqus/Explicit
- Continuum vs. Structural Elements
- Workshop 3: 3-D Solid Elements (IA)
- Workshop 3: 3-D Solid Elements (KW)

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

(E) Abaqus/Explicit only

1 hour
Lesson content:

- Overview
- Introduction
- Structural Elements in Abaqus
- Classical Formulation of Thin Shells and Slender Beams (S)
- Shear Flexible Formulation of Shells and Beams
- Thickness Changes
- Mode-based and Implicit Dynamics (S)

(S) Abaqus/Standard only
Lesson 7: Conventional Shell Elements

Lesson content:

- Conventional vs. Continuum Shell Elements
- Defining Conventional Shells
- Conventional Shell Element Behavior
- Conventional shell Element Types
- Comparison Studies
- Axisymmetric shell elements
- Viewing Shell Element Output
- Workshop 4: Shell Elements (IA)
- Workshop 4: Shell Elements (KW)

Both interactive (IA) and keywords (KW) versions of the workshop are provided. Complete only one.
Lesson 8: Continuum Shell Elements

Lesson content:

- Introduction to Continuum Shell Elements
- Defining the Thickness Direction for Continuum Shell Elements
- Continuum Shell Modeling
- Continuum Shell Example
- Shell Element Selection Summary
- Abaqus/CAE Demonstration: Controlling the Mesh Stack Direction

1 hour
Lesson 9: Beam and Frame Elements

Lesson content:

- Beam Elements
- Defining Beam Elements
- Viewing Beam Element Output
- Meshed Beam Cross Sections (S)
- Beam Element Selection Summary
- Frame Elements (S)
- Workshop 5: Beam Elements (IA)
- Workshop 5: Beam Elements (KW)

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