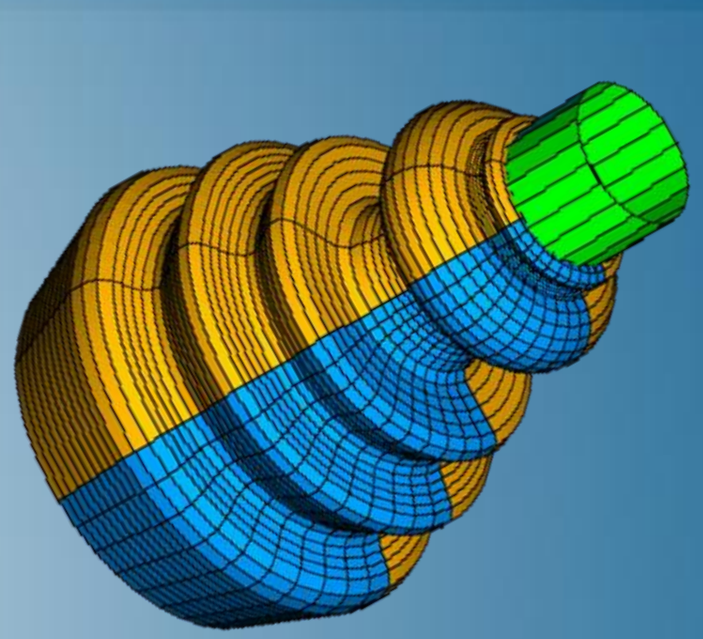
Element Selection in Abaqus

Abaqus 2018



3DEXPERIENCE®

About this Course

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

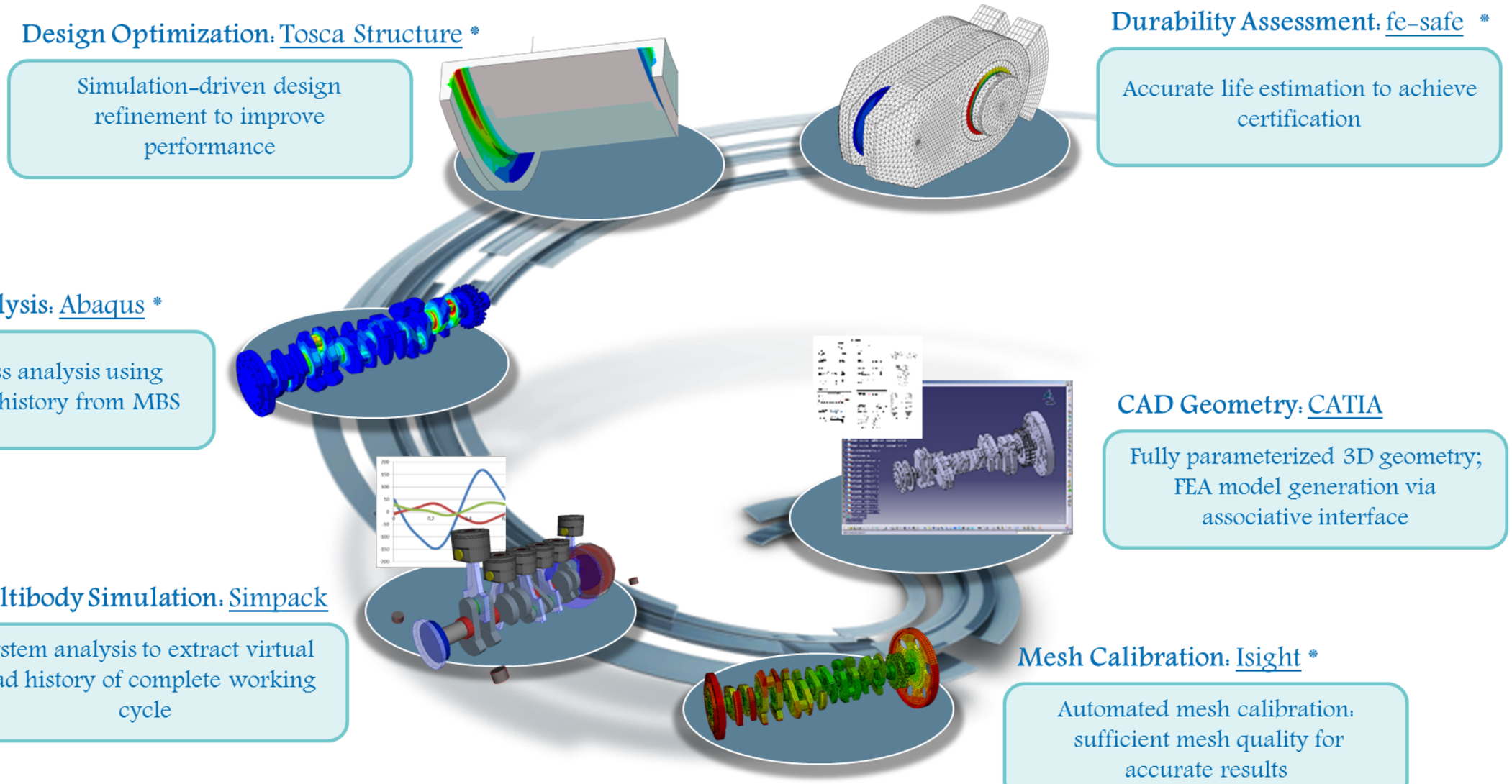
- Lecture 1 — Solid Elements in Abaqus
- Lecture 2 — Other Solid Element Types (*optional*)
- Lecture 3 — Integration, Hourglassing, & Incompressibility
  - Demo 1 — Hourglass control
  - Workshop 1 — Plane Strain Elements
- Lecture 4 — Key Properties of Solid Elements
  - Workshop 2 — Plane Stress Elements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

- SIMULIA is the Dassault Systèmes brand for Realistic Simulation solutions
- Portfolio of established, best-in-class products
  - Abaqus, Isight, Tosca, fe-safe, Simpack



Design Optimization: Tosca Structure *

Simulation-driven design refinement to improve performance

Durability Assessment: fe-safe *

Accurate life estimation to achieve certification

FEA Stress Analysis: Abaqus *

Detailed stress analysis using extracted load history from MBS

CAD Geometry: CATIA

Fully parameterized 3D geometry; FEA model generation via associative interface

Multibody Simulation: Simpack

System analysis to extract virtual load history of complete working cycle

Mesh Calibration: Isight *

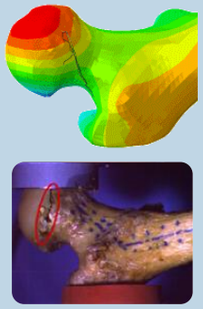
Automated mesh calibration: sufficient mesh quality for accurate results

* Included in extended licensing pool

SIMULIA's Power of the Portfolio

Abaqus

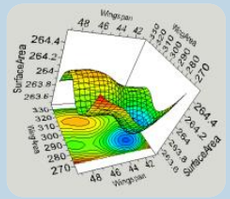
- Routine and Advanced Simulation
- Linear and Nonlinear, Static and Dynamic
- Thermal, Electrical, Acoustics
- Extended Physics through Co-simulation
- Model Preparation and Visualization



Realistic Human Simulation
High Speed Crash & Impact
Noise & Vibration

Isight

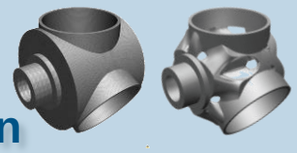
- Process Integration
- Design Optimization
- Parametric Optimization
- Six Sigma and Design of Experiments



Material Calibration
Workflow Automation
Design Exploration

Tosca

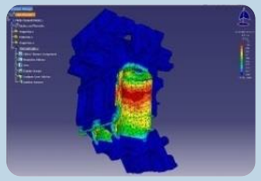
- Non-Parametric Optimization
- Structural and Fluid Flow Optimization
- Topology, Sizing, Shape, Bead Optimization



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

fe-safe

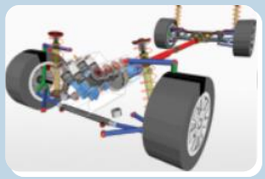
- Durability Simulation
- Low Cycle and High Cycle Fatigue
- Weld, High Temperature, Non-metallics



Safety Factors
Creep-Fatigue Interaction
Weld Fatigue

Simpack

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



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

Let the SIMULIA Learning Community be *Your* Portal to 21st Century Innovation

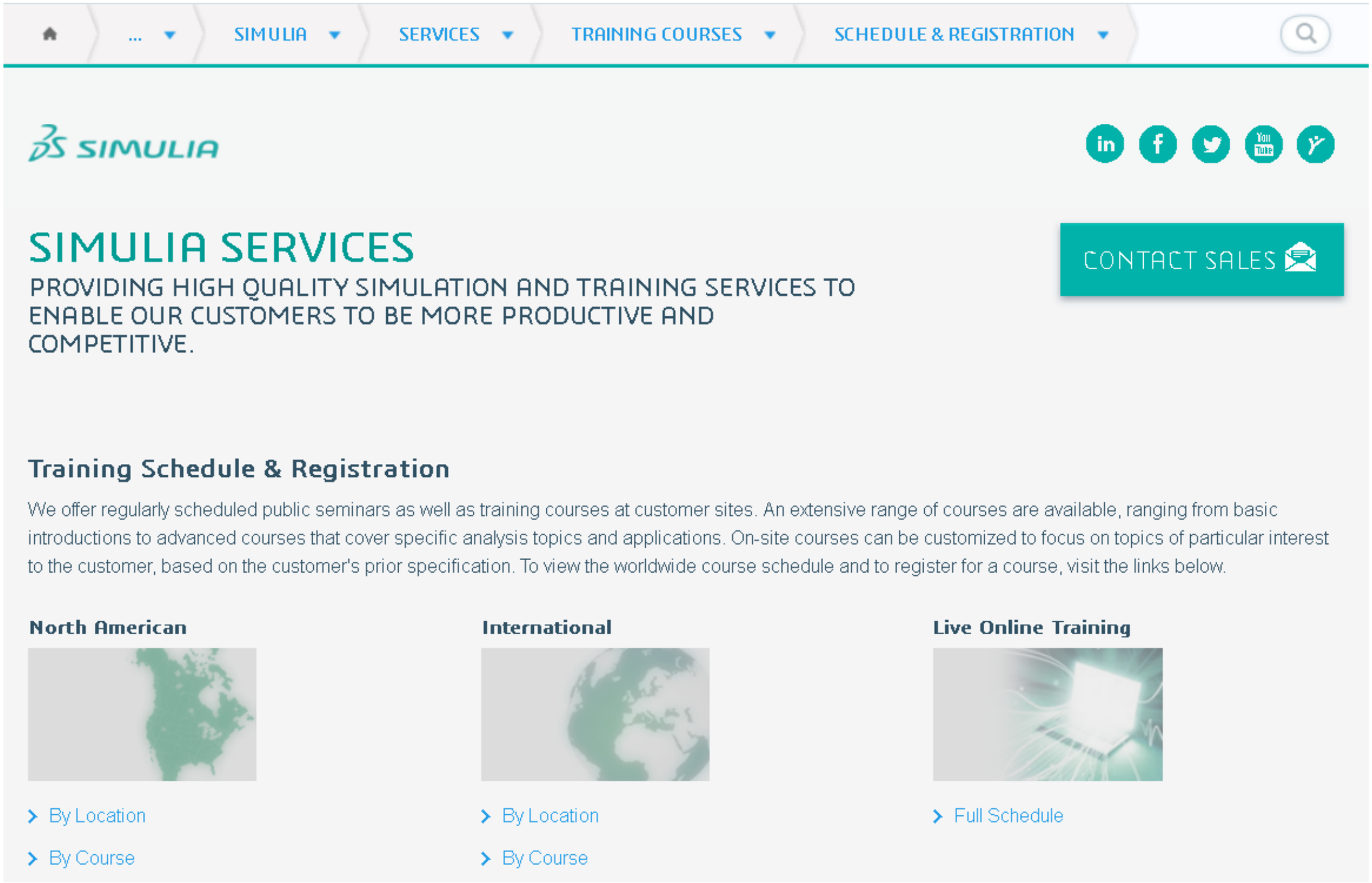
Discover new ways to explore how to leverage realistic simulation to drive product innovation. Join the thousands of Abaqus and Isight users who are already gaining valuable knowledge from the SIMULIA Learning Community.

For more information and registration, visit **3ds.com/simulia-learning**.
Connect. Share. Spark Innovation.

DASSAULT SYSTEMES | The **3D**EXPERIENCE Company

SIMULIA Training

http://www.3ds.com/products-services/simulia/services/training-courses/



Legal Notices

Revision Status

Lecture 1	11/17	Updated for Abaqus 2018
Lecture 2	11/17	Updated for Abaqus 2018
Lecture 3	11/17	Updated for Abaqus 2018
Lecture 4	11/17	Updated for Abaqus 2018
Lecture 5	11/17	Updated for Abaqus 2018
Lecture 6	11/17	Updated for Abaqus 2018
Lecture 7	11/17	Updated for Abaqus 2018
Lecture 8	11/17	Updated for Abaqus 2018
Lecture 9	11/17	Updated for Abaqus 2018
Demo 1	11/17	Updated for Abaqus 2018
Workshop 1	11/17	Updated for Abaqus 2018
Workshop 2	11/17	Updated for Abaqus 2018
Workshop 3	11/17	Updated for Abaqus 2018
Workshop 4	11/17	Updated for Abaqus 2018
Workshop 5	11/17	Updated for Abaqus 2018

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 3: Integration, Hourglassing, & Incompressibility

Lesson content:

(S) Abaqus/Standard only

- Numerical Integration
- Hourglassing and Hourglass Control
- Incompressible Material Behavior
- The $\overline{\boldsymbol{B}}$ 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.

3 hours

Lesson 4: Key Properties of Solid Elements

Lesson content:

(S) Abaqus/Standard only

- 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.

1 hour

Lesson 5: Modeling Bending and Stress Concentrations

Lesson content:

(E) Abaqus/Explicit only

- 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.

1 hour

Lesson 6: Structural Elements in Abaqus

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

1 hour

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.

2 hours

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:

(S) Abaqus/Standard only

- 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.

2 hours