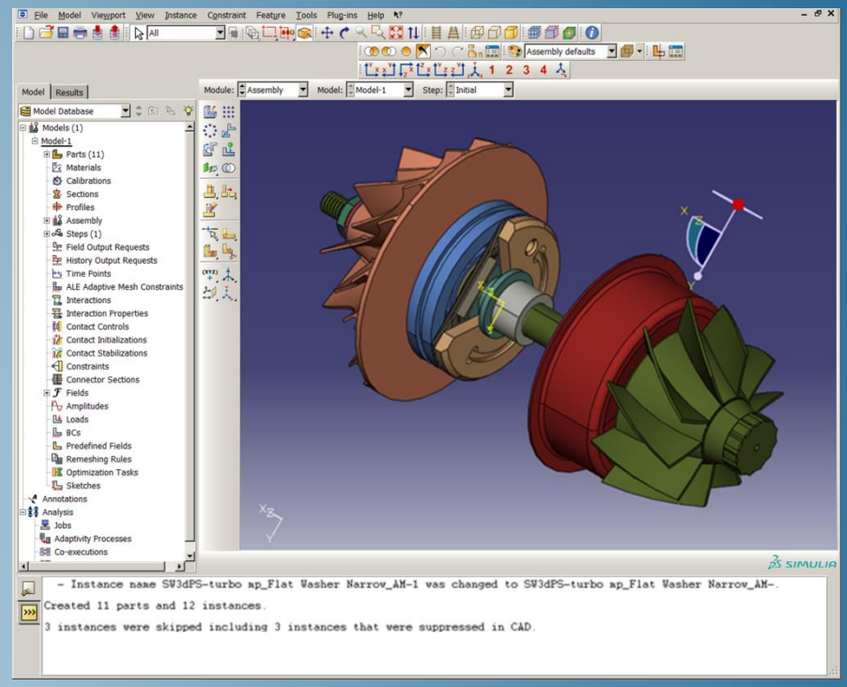
Introduction to Abaqus/CAE

Abaqus 2019



3DEXPERIENCE®

About this Course

Course objectives

Upon completion of this course you will be able to:

- Use Abaqus/CAE to create complete finite element models.
- Use Abaqus/CAE to submit and monitor analysis jobs.
- Use Abaqus/CAE to view and evaluate simulation results.

Targeted audience

Simulation Analysts

Prerequisites

None

2 days

Day 1

Lesson 1	Introducing Abaqus/CAE
Demo 1	**A First Look at Abaqus**
Workshop 1	**Overview of Abaqus/CAE**
Lesson 2	Working with Geometry in Abaqus/CAE
Demo 2a	**Working with Native Geometry**
Demo 2b	**Generating a Shell Feature from a Solid Feature**
Demo 2c	**Generating a Shell From a Thin Solid**
Workshop 2a	**Solid and Rigid Parts—Hinge Model**
Workshop 2b	**Creating Parts Using Constraints and Dimensions**
Workshop 2c	**Creating Parts—Clip and Plate Model**
Lesson 3	Working with Models Created Outside Abaqus
Demo 3	**Importing and Editing an Orphan Mesh**
Workshop 3	**Orphan Mesh Editing—Pump Model**

Day 2

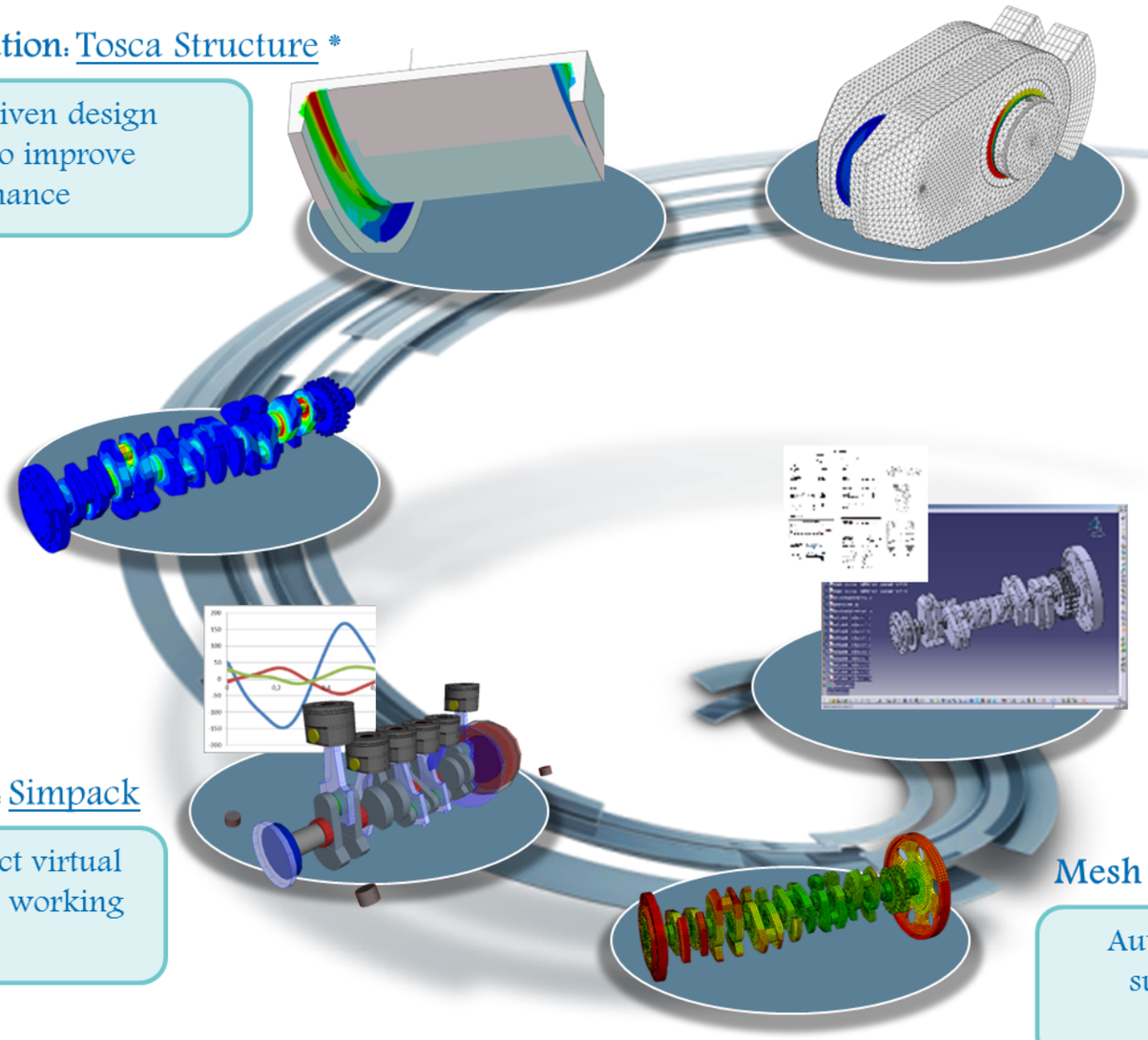
Lesson 4	Material Properties and Assemblies
Demo 4a	**Materials and Sections; Visualizing Beam Cross-Sections**
Demo 4b	**Creating an Assembly; Boolean Operations**
Workshop 4a	**Assigning Material Properties and Defining the Assembly—Hinge Model**
Workshop 4b	**Assigning Material Properties and Defining the Assembly—Clip and Plate Model**
Lesson 5	Steps, Interactions, and Loads
Demo 5a	**Creating Steps**
Demo 5b	**Using the Load Module**
Demo 5c	**Defining a Rigid Body**
Demo 5d	**Using Automatic Contact Detection and General Contact**
Workshop 5a	**Defining Steps, Interactions, Boundary Conditions, and Loads—Hinge Model**
Workshop 5b	**Defining Steps, Interactions, Boundary Conditions, and Loads—Clip and Plate Model**

Day 2 (cont'd)

Lesson 6	Meshing
Demo 6a	**Using the Mesh Module**
Demo 6b	**Partitioning and Meshing**
Workshop 6a	**Three-Dimensional Meshing—Hinge Model**
Workshop 6b	**Two-Dimensional Meshing—Fuse Model**
Workshop 6c	**Meshing—Clip and Plate Model**
Lesson 7	Job Management and Results Visualization
Demo 7a	**Using the Keywords Editor**
Demo 7b	**Visualizing Results**
Workshop 7a	**Job Management and Visualization—Hinge Model**
Workshop 7b	**Visualizing Results and Modifying the Model Definition—Clip and Plate Model**

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 *

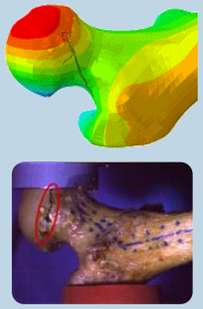
Automated mesh calibration: sufficient mesh quality for accurate results

* Included in extended licensing pool

SIMULIA's Power of the Portfolio

Abaqus

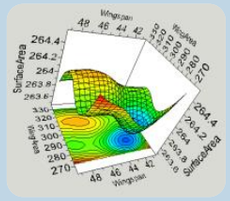
- Routine and Advanced Simulation
- Linear and Nonlinear, Static and Dynamic
- Thermal, Electrical, Acoustics
- Extended Physics through Co-simulation
- Model Preparation and Visualization



Realistic Human Simulation
High Speed Crash & Impact
Noise & Vibration

Isight

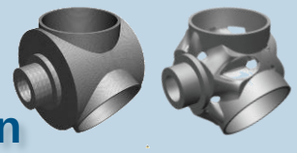
- Process Integration
- Design Optimization
- Parametric Optimization
- Six Sigma and Design of Experiments



Material Calibration
Workflow Automation
Design Exploration

Tosca

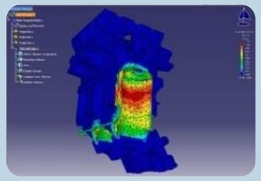
- Non-Parametric Optimization
- Structural and Fluid Flow Optimization
- Topology, Sizing, Shape, Bead Optimization



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

fe-safe

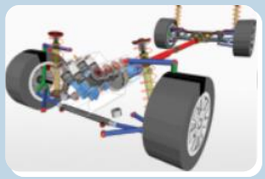
- Durability Simulation
- Low Cycle and High Cycle Fatigue
- Weld, High Temperature, Non-metallics



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

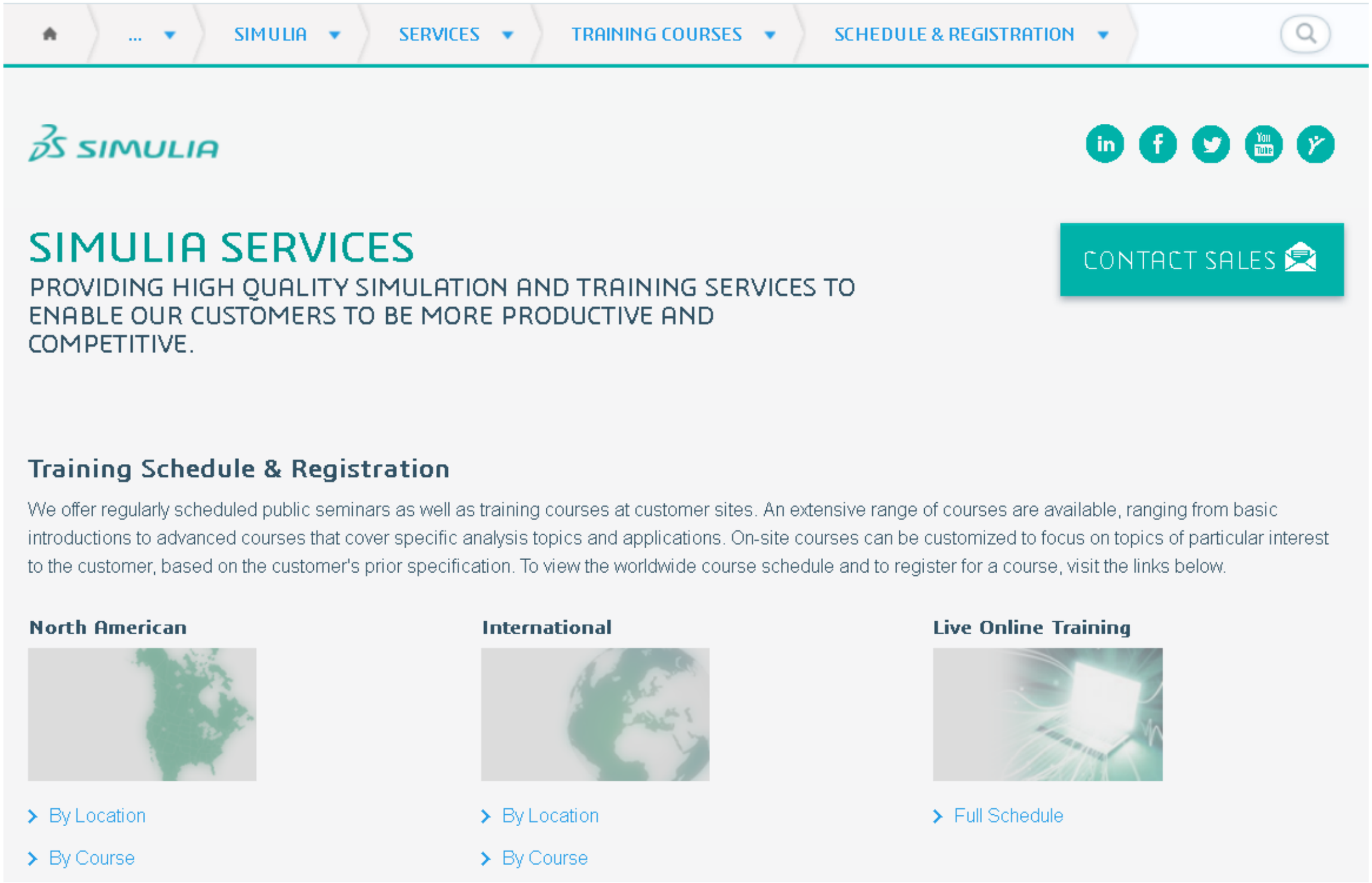
Discover new ways to explore how to leverage realistic simulation to drive product innovation. Join the thousands of Abaqus and Isight users who are already gaining valuable knowledge from the SIMULIA Learning Community.

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

Lesson 1	11/18	Updated for Abaqus 2019
Lesson 2	11/18	Updated for Abaqus 2019
Lesson 3	11/18	Updated for Abaqus 2019
Lesson 4	11/18	Updated for Abaqus 2019
Lesson 5	11/18	Updated for Abaqus 2019
Lesson 6	11/18	Updated for Abaqus 2019
Lesson 7	11/18	Updated for Abaqus 2019
Demonstration 1	11/18	Updated for Abaqus 2019
Demonstration 2a	11/18	Updated for Abaqus 2019
Demonstration 2b	11/18	Updated for Abaqus 2019
Demonstration 2c	11/18	Updated for Abaqus 2019
Demonstration 3	11/18	Updated for Abaqus 2019
Demonstration 4a	11/18	Updated for Abaqus 2019
Demonstration 4b	11/18	Updated for Abaqus 2019
Demonstration 5a	11/18	Updated for Abaqus 2019
Demonstration 5b	11/18	Updated for Abaqus 2019
Demonstration 5c	11/18	Updated for Abaqus 2019
Demonstration 5d	11/18	Updated for Abaqus 2019

Demonstration 6a	11/18	Updated for Abaqus 2019
Demonstration 6b	11/18	Updated for Abaqus 2019
Demonstration 7a	11/18	Updated for Abaqus 2019
Demonstration 7b	11/18	Updated for Abaqus 2019
Workshop 1	11/18	Updated for Abaqus 2019
Workshop 2a	11/18	Updated for Abaqus 2019
Workshop 2b	11/18	Updated for Abaqus 2019
Workshop 2c	11/18	Updated for Abaqus 2019
Workshop 3	11/18	Updated for Abaqus 2019
Workshop 4a	11/18	Updated for Abaqus 2019
Workshop 4b	11/18	Updated for Abaqus 2019
Workshop 5a	11/18	Updated for Abaqus 2019
Workshop 5b	11/18	Updated for Abaqus 2019
Workshop 6a	11/18	Updated for Abaqus 2019
Workshop 6b	11/18	Updated for Abaqus 2019
Workshop 6c	11/18	Updated for Abaqus 2019
Workshop 7a	11/18	Updated for Abaqus 2019
Workshop 7b	11/18	Updated for Abaqus 2019

Lesson 1: Introducing Abaqus/CAE

Lesson content:

- What is Abaqus FEA?
- Why Abaqus/CAE Over Other Tools?
- Primary Features of Abaqus/CAE
- Miscellaneous Features of Abaqus/CAE
- Starting Abaqus/CAE
- Important Abaqus/CAE Terminology
- Documentation
- SIMULIA Learning Community
- Abaqus Environment Settings
- Abaqus/CAE Checklist
- Working with the Model Tree
- Workshop Preliminaries
- Demonstration 1: A First Look at Abaqus/CAE
- Workshop 1: Overview of Abaqus/CAE

2 hours

Lesson 2: Working with Geometry in Abaqus/CAE

Lesson content:

- What are Parts?
- Defining a Part
- Geometry Import and Repair
- Building a Part Using the Part Module Tools
- Demonstration 2a: Working with Native Geometry
- Demonstration 2b: Generating a Shell Feature from a Solid Feature
- Demonstration 2c: Generating a Shell From a Thin Solid
- Workshop 2a: Solid and Rigid Parts—Hinge Model
- Workshop 2b: Creating Parts Using Constraints and Dimensions
- Workshop 2c: Creating Parts—Clip and Plate Model

3 hours

Lesson 3: Working with Models Created Outside Abaqus

Lesson content:

- Details of an Abaqus Input File
- Orphan Mesh Import
- Example
- Creating Geometry from an Orphan Mesh
- Combining Orphan and Native Mesh Features
- Demonstration 3: Importing and Editing an Orphan Mesh
- Workshop 3: Orphan Mesh Editing—Pump Model

2 hours

Lesson 4: Material Properties and Assemblies

Lesson content:

- Defining and Assigning Properties
- Material Evaluation
- Material Databases
- Material Calibration
- What is an Assembly?
- Positioning Instances
- Patterning
- Boolean Operations
- Sets and Surfaces
- Display Groups
- Hiding/Showing Instances
- Switching Context for Model and Part Instances
- Demonstration 4a: Materials and Sections; Visualizing Beam Cross-Sections
- Demonstration 4b: Creating an Assembly; Boolean Operations
- Workshop 4a: Assigning Material Properties and Defining the Assembly—Hinge Model
- Workshop 4b: Assigning Material Properties and Defining the Assembly—Clip and Plate Model

2 hours

Lesson 5: Steps, Interactions, and Loads

Lesson content:

- Steps
- Output
- Interactions
- Loads, Boundary Conditions, and Initial Conditions
- Model Verification
- Demonstration 5a: Creating Steps
- Demonstration 5b: Using the Load Module
- Demonstration 5c: Defining a Rigid Body
- Demonstration 5d: Using Automatic Contact Detection and General Contact
- Workshop 5a: Defining Steps, Interactions, Boundary Conditions, and Loads—Hinge Model
- Workshop 5b: Defining Steps, Interactions, Boundary Conditions, and Loads—Clip and Plate Model

2 hours

Lesson 6: Meshing

Lesson content:

- Overview
- Introduction
- Dependent and Independent Part Instances
- Mesh Generation Techniques
- Enabling Various Meshing Techniques
- Bottom-Up Hex Meshing
- Mesh Compatibility
- Controlling Mesh Density and Gradation
- Parametric Modeling
- Assigning Element Types
- Verifying Mesh Quality
- Mass and Mesh Queries
- Demonstration 6a: Using the Mesh Module
- Demonstration 6b: Partitioning and Meshing
- Workshop 6a: Three-Dimensional Meshing—Hinge Model
- Workshop 6b: Two-Dimensional Meshing—Fuse Model
- Workshop 6c: Meshing—Clip and Plate Model

2 hours

Lesson 7: Job Management and Results Visualization

Lesson content:

- Job Management
- Keywords Editor
- Review of ODB Output
- Results Visualization: Basic features
- Results Visualization: Advanced/miscellaneous features
- Demonstration 7a: Using the Keywords Editor
- Demonstration 7b: Visualizing Results
- Workshop 7a: Job Management and Visualization—Hinge Model
- Workshop 7b: Visualizing Results and Modifying the Model Definition—Clip and Plate Model

2 hours