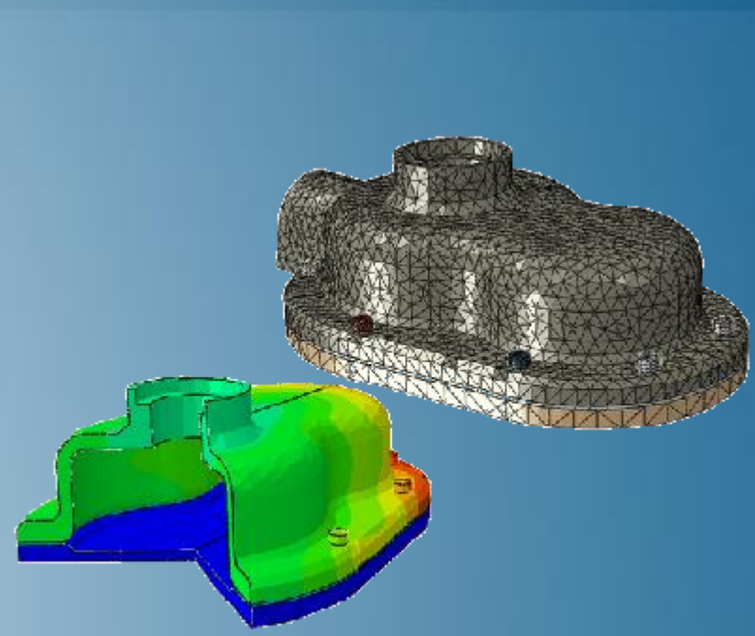
Introduction to Abaqus

Abaqus 2020



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.
- Solve structural analysis problems using Abaqus/Standard and Abaqus/Explicit, including the effects of material nonlinearity, large deformation and contact.

Targeted audience

Simulation Analysts

Prerequisites

None

4 days

Day 1

Lesson 1	Overview of Abaqus
Demo 1	**A First Look at Abaqus**
Workshop 1	**Linear Static Analysis of a Cantilever Beam**
Lesson 2	Working with Geometry (Part 1)
Demo 2	**Working with Native Geometry**
Workshop 2	**Creating Native Geometry: Pipe Creep Model**
Lesson 3	Working with Geometry (Part 2)
Demo 3a	**Generating a Shell From a Thin Solid**
Workshop 3a	**Import and Geometry Repair of Intersecting Pipes**
Demo 3b	**Importing and Editing an Orphan Mesh**
Workshop 3b	**Importing and Editing an Orphan Mesh: Pump Model**

Day 2

Lesson 4	Material and Section Properties
Demo 4	**Creating Materials and Assigning Sections**
Workshop 4a	**Material and Section Properties: Pipe Creep Model**
Workshop 4b	**Material and Section Properties: Pump Model**
Lesson 5	Assemblies in Abaqus
Demo 5	**Creating an Assembly**
Workshop 5	**Pump Model Assembly**
Lesson 6	Steps, Output, Loads, & Boundary Conditions
Demo 6a	**Creating Steps**
Demo 6b	**Using the Load Module**
Workshop 6a	**Step Definition and Loads: Pipe Creep Model**
Workshop 6b	**Step Definition and Loads: Pump Model**
Lesson 7	Meshing Imported and Native Geometry
Demo 7	**Using the Mesh Module**
Workshop 7a	**Structured Hex Meshing: Pipe Creep Model**
Workshop 7b	**Free and Swept Meshing: Pump Model**
Workshop 7c	**Meshing of Intersecting Pipes**

Day 3

Lesson 8	Job Management and Results Visualization
Demo 8a	**Using the Keywords Editor**
Demo 8b	**Visualizing Results**
Workshop 8	**Creep of a Pipe Intersection**
Lesson 9	Linear and Nonlinear Problems
Lesson 10	Analysis Procedures (Part 1)
Demo 10	**Nonlinear Static Analysis**
Workshop 10a	**Linear Analysis of a Skew Plate**
Workshop 10b	**Nonlinear Analysis of a Skew Plate**
Lesson 11	Analysis Procedures (Part 2)
Demo 11	**Multiple Load Cases**
Workshop 11	**Linear Static Analysis of a Cantilever Beam** (***optional***)

Day 4

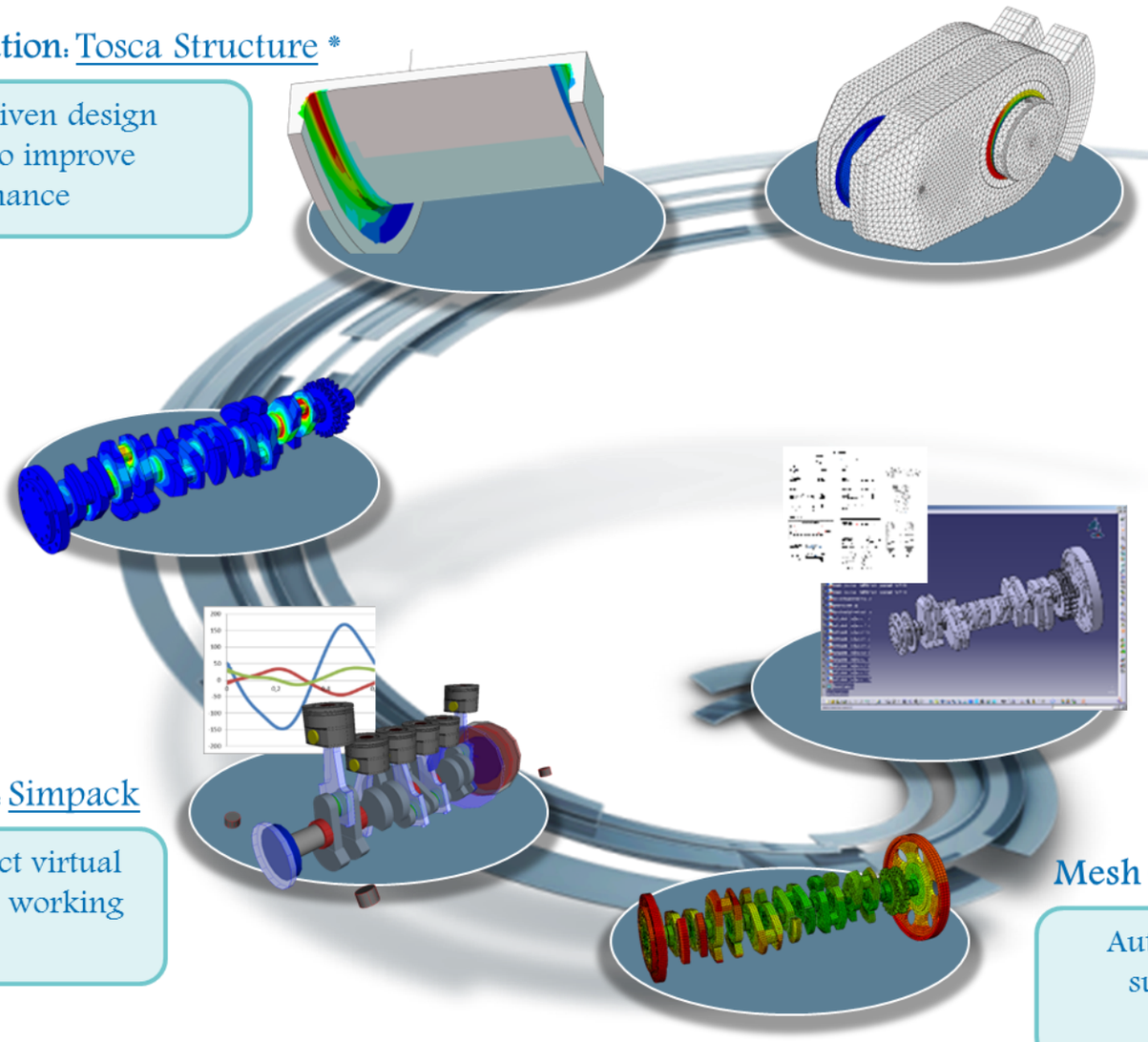
Lesson 12	Analysis Procedures (Part 3)
Workshop 12a	**Dynamic Analysis of a Skew Plate**
Workshop 12b	**Pipe Whip Analysis**
Lesson 13	Analysis Continuation Techniques
Workshop 13	**Unloading Analysis of a Skew Plate**
Lesson 14	Constraints and Connections
Demo 14	**Defining a Rigid Body**
Workshop 14	**Tie Constraints: Pump Model**
Lesson 15	Contact
Demo 15	**Using Automatic Contact Detection and General Contact**
Workshop 15	**Nonlinear Static Analysis of a Pump Assembly**

Additional Material

Appendix 1	Element Selection Criteria
Appendix 2	Analyzing Highly Nonlinear Quasi-Static Problems
Workshop A2	**Single Pass Rolling of a Thick Plate**
Appendix 3	Heat Transfer and Thermal-Stress Analysis
Workshop A3	**Thermal-Stress Analysis of Intersecting Pipes**

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 *

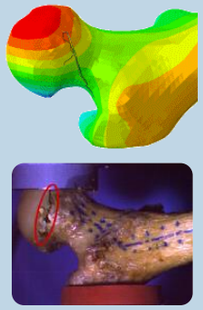
Automated mesh calibration: sufficient mesh quality for accurate results

* Included in extended licensing pool

SIMULIA's Power of the Portfolio

Abaqus

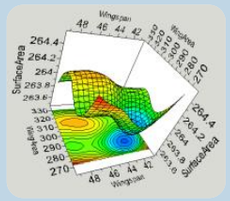
- Routine and Advanced Simulation
- Linear and Nonlinear, Static and Dynamic
- Thermal, Electrical, Acoustics
- Extended Physics through Co-simulation
- Model Preparation and Visualization



Realistic Human Simulation
High Speed Crash & Impact
Noise & Vibration

Isight

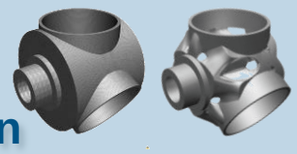
- Process Integration
- Design Optimization
- Parametric Optimization
- Six Sigma and Design of Experiments



Material Calibration
Workflow Automation
Design Exploration

Tosca

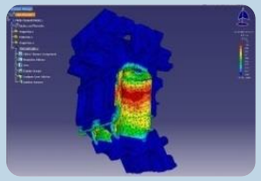
- Non-Parametric Optimization
- Structural and Fluid Flow Optimization
- Topology, Sizing, Shape, Bead Optimization



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

fe-safe

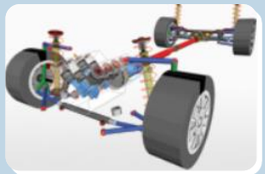
- Durability Simulation
- Low Cycle and High Cycle Fatigue
- Weld, High Temperature, Non-metallics



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

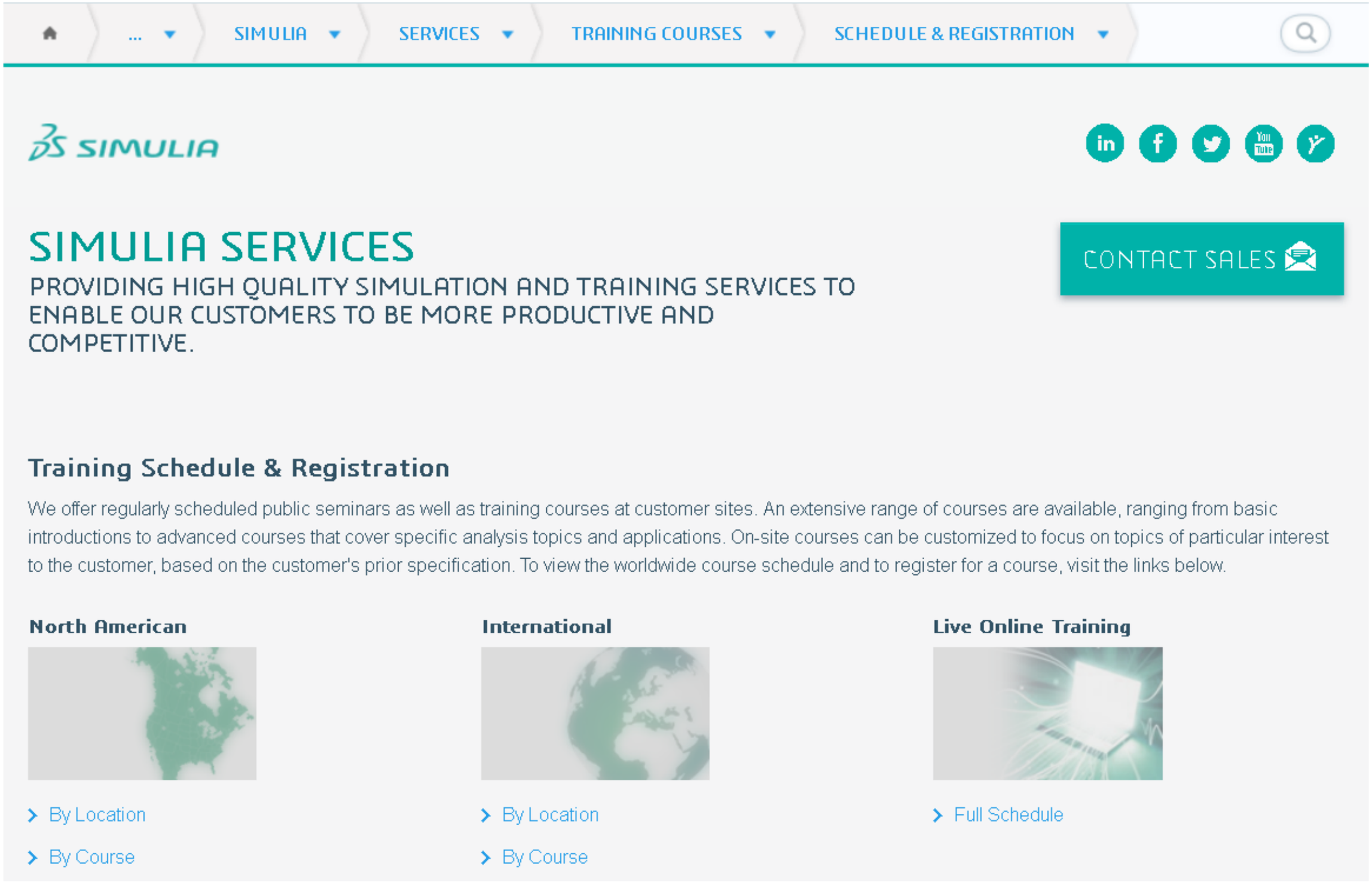
Discover new ways to explore how to leverage realistic simulation to drive product innovation. Join the thousands of Abaqus and Isight users who are already gaining valuable knowledge from the SIMULIA Learning Community.

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 (1/2)

Lesson 1	11/19	Updated for Abaqus 2020
Lesson 2	11/19	Updated for Abaqus 2020
Lesson 3	11/19	Updated for Abaqus 2020
Lesson 4	11/19	Updated for Abaqus 2020
Lesson 5	11/19	Updated for Abaqus 2020
Lesson 6	11/19	Updated for Abaqus 2020
Lesson 7	11/19	Updated for Abaqus 2020
Lesson 8	11/19	Updated for Abaqus 2020
Lesson 9	11/19	Updated for Abaqus 2020
Lesson 10	11/19	Updated for Abaqus 2020
Lesson 11	11/19	Updated for Abaqus 2020
Lesson 12	11/19	Updated for Abaqus 2020
Lesson 13	11/19	Updated for Abaqus 2020
Lesson 14	11/19	Updated for Abaqus 2020
Lesson 15	11/19	Updated for Abaqus 2020
Appendix 1	11/19	Updated for Abaqus 2020
Appendix 2	11/19	Updated for Abaqus 2020
Appendix 3	11/19	Updated for Abaqus 2020

Demonstration 1	11/19	Updated for Abaqus 2020
Demonstration 2	11/19	Updated for Abaqus 2020
Demonstration 3a	11/19	Updated for Abaqus 2020
Demonstration 3b	11/19	Updated for Abaqus 2020
Demonstration 4	11/19	Updated for Abaqus 2020
Demonstration 5	11/19	Updated for Abaqus 2020
Demonstration 6a	11/19	Updated for Abaqus 2020
Demonstration 6b	11/19	Updated for Abaqus 2020
Demonstration 7	11/19	Updated for Abaqus 2020
Demonstration 8a	11/19	Updated for Abaqus 2020
Demonstration 8b	11/19	Updated for Abaqus 2020
Demonstration 10	11/19	Updated for Abaqus 2020
Demonstration 11	11/19	Updated for Abaqus 2020
Demonstration 14	11/19	Updated for Abaqus 2020
Demonstration 15	11/19	Updated for Abaqus 2020

Revision Status (2/2)

Workshop 1	11/19	Updated for Abaqus 2020
Workshop 2	11/19	Updated for Abaqus 2020
Workshop 3a	11/19	Updated for Abaqus 2020
Workshop 3b	11/19	Updated for Abaqus 2020
Workshop 4a	11/19	Updated for Abaqus 2020
Workshop 4b	11/19	Updated for Abaqus 2020
Workshop 5	11/19	Updated for Abaqus 2020
Workshop 6a	11/19	Updated for Abaqus 2020
Workshop 6b	11/19	Updated for Abaqus 2020
Workshop 7a	11/19	Updated for Abaqus 2020
Workshop 7b	11/19	Updated for Abaqus 2020
Workshop 7c	11/19	Updated for Abaqus 2020
Workshop 8	11/19	Updated for Abaqus 2020

Workshop 10a	11/19	Updated for Abaqus 2020
Workshop 10b	11/19	Updated for Abaqus 2020
Workshop 11	11/19	Updated for Abaqus 2020
Workshop 12a	11/19	Updated for Abaqus 2020
Workshop 12b	11/19	Updated for Abaqus 2020
Workshop 13	11/19	Updated for Abaqus 2020
Workshop 14	11/19	Updated for Abaqus 2020
Workshop 15	11/19	Updated for Abaqus 2020
Workshop A2	11/19	Updated for Abaqus 2020
Workshop A3	11/19	Updated for Abaqus 2020

Lesson 1: Overview of Abaqus

Lesson content:

- What is Abaqus FEA?
- Abaqus/CAE
- Abaqus/Standard and Abaqus/Explicit
- Abaqus Conventions
- Working with the Model Tree
- Other Abaqus/CAE Topics
- Documentation
- Learning Community
- Abaqus Environment Settings
- Abaqus Fetch Utility
- Workshop Preliminaries
- Demonstration 1: A First Look at Abaqus/CAE
- Workshop 1: Linear Static Analysis of a Cantilever Beam

2.5 hours

Lesson 2: Working with Geometry (Part 1)

Lesson content:

- Abaqus/CAE: Part Module
- What are Parts?
- Creating Part Geometry
- Building a Part Using the Part Module Tools
- The Sketcher
- Adding Features
- Miscellaneous Topics
- Demonstration 2: Working with Native Geometry
- Workshop 2: Creating Native Geometry: Pipe Creep Model

2 hours

Lesson 3: Working with Geometry (Part 2)

Lesson content:

- Abaqus/CAE: Part Module
- Geometry Import and Repair
- Demonstration 3a: Generating a Shell From a Thin Solid
- Workshop 3a: Geometry Repair of Intersecting Pipes
- Part from an Orphan Mesh
- Creating Geometry from an Orphan Mesh
- Demonstration 3b: Importing and Editing an Orphan Mesh
- Workshop 3b: Importing and Editing an Orphan Mesh: Pump Model

1 hour

Lesson 4: Material and Section Properties

Lesson content:

- Abaqus/CAE: Property Module
- Abaqus Material Definitions
- Abaqus Conventions
- Linear Elasticity
- Large Strain Elasticity
- Metal Plasticity
- Material Calibration
- Material Databases
- Section Properties
- Special Features: Skins and Stringers
- Demonstration 4: Creating Materials and Assigning Sections
- Workshop 4a: Material and Section Properties: Pipe Model
- Workshop 4b: Material and Section Properties: Pump Model

2 hours

Lesson 5: Assemblies in Abaqus

Lesson content:

- Abaqus/CAE: Assembly Module
- What is an Assembly?
- Positioning Instances
- Subassemblies
- Other Operations
- Sets
- Surfaces
- Display Groups
- Instance Types
- Demonstration 5: Creating an Assembly; Boolean Operations
- Workshop 5: Pump Model Assembly

2 hours

Lesson 6: Steps, Output, Loads, & Boundary Conditions

Lesson content:

- Abaqus/CAE: Step Module
- Analysis Steps and Procedures
- Demonstration 6a: Creating Steps
- Output Requests
- Output Files
- Abaqus/CAE: Load Module
- Amplitudes and Distributions
- Loads and Boundary Conditions
- Initial Conditions
- Demonstration 6b: Using the Load Module
- Workshop 6a: Step Definition and Loads: Pipe Creep Model
- Workshop 6b: Step Definition and Loads: Pump Model

2 hours

Lesson 7: Meshing Imported and Native Geometry

Lesson content:

- Abaqus/CAE: Mesh Module
- What is a Mesh?
- Elements in Abaqus
- Mesh Generation Workflow
- The Mesh Module
- Common Tools:
  - Density
  - Controls
  - Element Selection
  - Meshing
  - Local Fine-tuning
  - Quality Checks
- Advanced Topics:
  - Virtual Topology
  - Bottom-up Meshing
  - Mesh Compatibility
  - Mesh Convergence
- Dependent and Independent Part Instances
- Demonstration 7: Using the Mesh Module
- Workshop 7a: Structured Hex Meshing: Pipe Creep Model
- Workshop 7b: Free and Swept Meshing: Pump Model
- Workshop 7c: Meshing of Intersecting Pipes

2 hours

Lesson 8: Job Management and Results Visualization

Lesson content:

- Abaqus/CAE: Job Module
- Analysis Jobs
- Creating a Job
- The Job Manager
- Monitoring the Progress of an Analysis
- Keywords Editor
- Demonstration 8a: Using the Keywords Editor
- Viewing and Interpreting Results
- Abaqus/CAE: Visualization Module
- Viewing and Interpreting Results
- Output
- Example 1: Overhead Hoist
- Example 2: Overhead Hoist – Dynamic Loading
- Example 3: Connecting Lug
- Additional Topics
  - Color Coding
  - Display Groups
  - Managing Viewports
  - Display Options
- Demonstration 8b: Visualizing Results
- Advanced Topics
  - Result Options
- Final Thoughts
- Workshop 8: Creep of a Pipe Intersection

3 hours

Lesson 9: Linear and Nonlinear Problems

Lesson content:

- Is my problem nonlinear?
- What are the main sources of nonlinearities?
- Why are nonlinear problems hard to solve?
- How are nonlinear problems solved?
- Summary

30 minutes

Lesson 10: Analysis Procedures (Part 1)

Lesson content:

- Preliminaries
  - Abaqus Model and Analysis Steps
  - Analysis Procedures
- The *static, general* analysis procedure
- Finding a *converged* solution
- Demonstration 10: Nonlinear Static Analysis
- Workshop 10a: Linear Analysis of a Skew Plate
- Workshop 10b: Nonlinear Analysis of a Skew Plate

2 hours

Lesson 11: Analysis Procedures (Part 2)

Lesson content:

- Preliminaries: Analysis Procedures
- Linear Perturbation Procedures
- The *Static, Linear Perturbation* procedure
- *Buckle* procedure
- *Frequency* procedure
- Summary, so far...
- Multistep Analyses
- Demonstration 11: Load Cases and Multi-Step analysis
- Workshop 11: Linear Static Analysis of a Cantilever Beam (*optional*)

2 hours

Lesson 12: Analysis Procedures (Part 3)

Lesson content:

- Preliminaries
  - Analysis Procedures
  - What Makes a Problem Dynamic?
  - Implicit vs Explicit time integration
- The *dynamic, explicit* analysis procedure
- Stability Limit
- Finding a solution … faster!
- Troubleshooting Abaqus/Explicit analyses
- Workshop 12a: Dynamic Analysis of a Skew Plate
- Workshop 12b: Pipe Whip Analysis

2 hours

Lesson 13: Analysis Continuation Techniques

Lesson content:

- Analysis Continuation Techniques
- Restarting an Abaqus Analysis
- Workshop 13: Unloading Analysis of a Skew Plate

30 minutes

Lesson 14: Constraints and Connections

Lesson content:

- Introduction
- Rigid Body Constraint
- Tie Constraint
- Coupling Constraint
- Shell-to-Solid Coupling
- Connector Elements
- Mesh-independent Fasteners
- Demonstration 14: Defining a Rigid Body
- Workshop 14: Tie Constraints: Pump Model

1 hour

Lesson 15: Contact

Lesson content:

- Introduction
- Mechanical Contact Properties
- Contact Domain
- Contact Formulation and Controls
- Summary
- Example 1: Shearing of a lap joint with contact pairs
- Example 2: Shearing of a lap joint with general contact
- Example 3: Crimp forming with general contact
- Additional Topics
  - Handling Initial Overclosures
  - Contact Output
  - Modeling Tips
- Demonstration 15: Using Automatic Contact Detection and General Contact
- Workshop 15: Nonlinear Static Analysis of a Pump Assembly

2 hours

Appendix 1: Element Selection Criteria

Appendix content:

- Elements
- Structural (Shells and Beams) vs. Continuum Elements
- Modeling Bending Using Continuum Elements
- Stress Concentrations
- Contact
- Incompressible Materials
- Mesh Generation
- Solid Element Selection Summary

1.5 hours

Appendix 2: Analyzing Nonlinear Quasi-Static Problems

Appendix content:

- Introduction
- Solution Strategies
- Quasi-Static Simulations Using Explicit Dynamics
- Adaptive Meshing
- Workshop A2: Single Pass Rolling of a Thick Plate

3 hours

Appendix 3: Heat Transfer and Thermal-Stress Analysis

Appendix content:

- Introduction
- Steady-State Heat Transfer
- Transient Heat Transfer
- Thermal Interfaces
- Thermal-Stress Analysis
- Workshop A3: Thermal-Stress Analysis of Intersecting Pipes

2 hours