Introduction to Abaqus

Abaqus 2020
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
<table>
<thead>
<tr>
<th>Lesson 4</th>
<th>Material and Section Properties</th>
</tr>
</thead>
<tbody>
<tr>
<td>Demo 4</td>
<td>Creating Materials and Assigning Sections</td>
</tr>
<tr>
<td>Workshop 4a</td>
<td>Material and Section Properties: Pipe Creep Model</td>
</tr>
<tr>
<td>Workshop 4b</td>
<td>Material and Section Properties: Pump Model</td>
</tr>
<tr>
<td>Lesson 5</td>
<td>Assemblies in Abaqus</td>
</tr>
<tr>
<td>Demo 5</td>
<td>Creating an Assembly</td>
</tr>
<tr>
<td>Workshop 5</td>
<td>Pump Model Assembly</td>
</tr>
<tr>
<td>Lesson 6</td>
<td>Steps, Output, Loads, &amp; Boundary Conditions</td>
</tr>
<tr>
<td>Demo 6a</td>
<td>Creating Steps</td>
</tr>
<tr>
<td>Demo 6b</td>
<td>Using the Load Module</td>
</tr>
<tr>
<td>Workshop 6a</td>
<td>Step Definition and Loads: Pipe Creep Model</td>
</tr>
<tr>
<td>Workshop 6b</td>
<td>Step Definition and Loads: Pump Model</td>
</tr>
<tr>
<td>Lesson 7</td>
<td>Meshing Imported and Native Geometry</td>
</tr>
<tr>
<td>Demo 7</td>
<td>Using the Mesh Module</td>
</tr>
<tr>
<td>Workshop 7a</td>
<td>Structured Hex Meshing: Pipe Creep Model</td>
</tr>
<tr>
<td>Workshop 7b</td>
<td>Free and Swept Meshing: Pump Model</td>
</tr>
<tr>
<td>Workshop 7c</td>
<td>Meshing of Intersecting Pipes</td>
</tr>
</tbody>
</table>
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 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

- FEA Stress Analysis, Abaqus *
  Detailed stress analysis using extracted load history from MBS

- Multibody Simulation, Simpack
  System analysis to extract virtual load history of complete working cycle

- Durability Assessment, fe-safe *
  Accurate life estimation to achieve certification

- CAD Geometry, CATIA
  Fully parameterized 3D geometry; FEA model generation via associative interface

- Mesh Calibration, Isight *
  Automated mesh calibration; sufficient mesh quality for accurate results

* Included in extended licensing pool
### SIMULIA's Power of the Portfolio

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

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.
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
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, 2019

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 (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)

<table>
<thead>
<tr>
<th>Workshop</th>
<th>Date</th>
<th>Updated for Abaqus 2020</th>
</tr>
</thead>
<tbody>
<tr>
<td>Workshop 1</td>
<td>11/19</td>
<td>Updated for Abaqus 2020</td>
</tr>
<tr>
<td>Workshop 2</td>
<td>11/19</td>
<td>Updated for Abaqus 2020</td>
</tr>
<tr>
<td>Workshop 3a</td>
<td>11/19</td>
<td>Updated for Abaqus 2020</td>
</tr>
<tr>
<td>Workshop 3b</td>
<td>11/19</td>
<td>Updated for Abaqus 2020</td>
</tr>
<tr>
<td>Workshop 4a</td>
<td>11/19</td>
<td>Updated for Abaqus 2020</td>
</tr>
<tr>
<td>Workshop 4b</td>
<td>11/19</td>
<td>Updated for Abaqus 2020</td>
</tr>
<tr>
<td>Workshop 5</td>
<td>11/19</td>
<td>Updated for Abaqus 2020</td>
</tr>
<tr>
<td>Workshop 6a</td>
<td>11/19</td>
<td>Updated for Abaqus 2020</td>
</tr>
<tr>
<td>Workshop 6b</td>
<td>11/19</td>
<td>Updated for Abaqus 2020</td>
</tr>
<tr>
<td>Workshop 7a</td>
<td>11/19</td>
<td>Updated for Abaqus 2020</td>
</tr>
<tr>
<td>Workshop 7b</td>
<td>11/19</td>
<td>Updated for Abaqus 2020</td>
</tr>
<tr>
<td>Workshop 7c</td>
<td>11/19</td>
<td>Updated for Abaqus 2020</td>
</tr>
<tr>
<td>Workshop 8</td>
<td>11/19</td>
<td>Updated for Abaqus 2020</td>
</tr>
<tr>
<td>Workshop 10a</td>
<td>11/19</td>
<td>Updated for Abaqus 2020</td>
</tr>
<tr>
<td>Workshop 10b</td>
<td>11/19</td>
<td>Updated for Abaqus 2020</td>
</tr>
<tr>
<td>Workshop 11</td>
<td>11/19</td>
<td>Updated for Abaqus 2020</td>
</tr>
<tr>
<td>Workshop 12a</td>
<td>11/19</td>
<td>Updated for Abaqus 2020</td>
</tr>
<tr>
<td>Workshop 12b</td>
<td>11/19</td>
<td>Updated for Abaqus 2020</td>
</tr>
<tr>
<td>Workshop 13</td>
<td>11/19</td>
<td>Updated for Abaqus 2020</td>
</tr>
<tr>
<td>Workshop 14</td>
<td>11/19</td>
<td>Updated for Abaqus 2020</td>
</tr>
<tr>
<td>Workshop 15</td>
<td>11/19</td>
<td>Updated for Abaqus 2020</td>
</tr>
<tr>
<td>Workshop A2</td>
<td>11/19</td>
<td>Updated for Abaqus 2020</td>
</tr>
<tr>
<td>Workshop A3</td>
<td>11/19</td>
<td>Updated for Abaqus 2020</td>
</tr>
</tbody>
</table>
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
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
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 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
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

Lesson 7: Meshing Imported and Native Geometry
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
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
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
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
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