Day 1

- **Lecture 1** Overview of Abaqus
  - **Demo 1** A First Look at Abaqus
  - **Workshop 1** Linear Static Analysis of a Cantilever Beam
- **Lecture 2** Working with Geometry in Abaqus
  - **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** Creating Native Geometry: Pipe Creep Model
  - **Workshop 2b** Import and Geometry Repair: Intersecting Pipes
- **Lecture 3** Working with Models Created Outside Abaqus
  - **Demo 3** Importing and Editing an Orphan Mesh
  - **Workshop 3** Importing and Editing an Orphan Mesh: Pump Model
Day 2

• Lecture 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
• Lecture 5  Assemblies in Abaqus
  • Demo 5  Creating an Assembly
  • Workshop 5  Pump Model Assembly
• Lecture 6  Steps, Output, Loads, and 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

Day 2 (cont’d)

• Lecture 7  Meshing Imported and Native Geometry
  • Demo 7a  Using the Mesh Module
  • Demo 7b  Partitioning and Meshing
  • Workshop 7a  Structured Hex Meshing: Pipe Creep Model
  • Workshop 7b  Free and Swept Meshing: Pump Model
  • Workshop 7c  Meshing of Intersecting Pipes
## Day 3

- **Lecture 8**  
  Job Management and Results Visualization
  - **Demo 8a**  
    Using the Keywords Editor  
  - **Demo 8b**  
    Visualizing Results  
  - **Workshop 8**  
    Creep of a Pipe Intersection

- **Lecture 9**  
  Linear Static Analysis
  - **Demo 9**  
    Multiple Load Cases  
  - **Workshop 9a**  
    Linear Static Analysis of a Cantilever Beam: Multiple Load Cases  
  - **Workshop 9b**  
    Linear Analysis of a Skew Plate

- **Lecture 10**  
  Adding Material and Geometric Nonlinearity
  - **Demo 10**  
    Nonlinear Static Analysis  
  - **Workshop 10**  
    Nonlinear Analysis of a Skew Plate

## Day 4

- **Lecture 11**  
  Multistep Analysis in Abaqus
  - **Workshop 11**  
    Unloading Analysis of a Skew Plate

- **Lecture 12**  
  Constraints and Connections
  - **Demo 12**  
    Defining a Rigid Body  
  - **Workshop 12**  
    Defining Tie Constraints: Pump Model

- **Lecture 13**  
  Contact
  - **Demo 13**  
    Using Automatic Contact Detection and General Contact  
  - **Workshop 13**  
    Nonlinear Static Analysis of a Pump Assembly
Day 5

- Lecture 14 Dynamic Analysis
  - Demo 14 Dynamic Analysis
  - Workshop 14a Dynamic Analysis of a Skew Plate
  - Workshop 14b Pipe Whip Analysis
- Lecture 15 Analyzing Highly Nonlinear Quasi-Static Problems
  - Demo 15 Quasi-Static Analysis
  - Workshop 15 Single Pass Rolling of a Thick Plate
- Lecture 16 Heat Transfer and Thermal-Stress Analysis
  - Demo 16 Thermal Analysis
  - Workshop 16 Thermal-Stress Analysis of Intersecting Pipes

Additional Material

- Appendix 1 Introduction to Finite Element Analysis
- Appendix 2 Element Selection Criteria
- Appendix 3 More on Contact
**Legal Notices**

The Abaqus Software described in this documentation is available only under license from Dassault Systèmes and its subsidiary 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 subsidiary.

© Dassault Systèmes, 2010.

Printed in the United States of America

Abaqus, the 3DS logo, SIMULIA and CATIA 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 Abaqus 6.10 Release Notes and the notices at: http://www.simulia.com/products/products_legal.html.

---

**Revision Status**

| Lecture 1  | 5/10 Updated for 6.10 |
| Lecture 2  | 5/10 Updated for 6.10 |
| Lecture 3  | 5/10 Updated for 6.10 |
| Lecture 4  | 5/10 Updated for 6.10 |
| Lecture 5  | 5/10 Updated for 6.10 |
| Lecture 6  | 5/10 Updated for 6.10 |
| Lecture 7  | 5/10 Updated for 6.10 |
| Lecture 8  | 5/10 Updated for 6.10 |
| Lecture 9  | 5/10 Updated for 6.10 |
| Lecture 10 | 5/10 Updated for 6.10 |
| Lecture 11 | 5/10 Updated for 6.10 |
| Lecture 12 | 5/10 Updated for 6.10 |
| Lecture 13 | 5/10 Updated for 6.10 |
| Lecture 14 | 5/10 Updated for 6.10 |
| Lecture 15 | 5/10 Updated for 6.10 |
| Lecture 16 | 5/10 Updated for 6.10 |
| Appendix 1 | 5/10 Updated for 6.10 |
| Appendix 2 | 5/10 Updated for 6.10 |
| Appendix 3 | 5/10 Updated for 6.10 |
| Demonstration 1 | 5/10 Updated for 6.10 |
| Demonstration 2a | 5/10 Updated for 6.10 |
| Demonstration 2b | 5/10 Updated for 6.10 |
| Demonstration 2c | 5/10 New for 6.10 |
| Demonstration 3 | 5/10 Updated for 6.10 |
| Demonstration 4 | 5/10 Updated for 6.10 |
| Demonstration 5 | 5/10 Updated for 6.10 |
| Demonstration 6a | 5/10 Updated for 6.10 |
| Demonstration 6b | 5/10 Updated for 6.10 |
| Demonstration 7a | 5/10 Updated for 6.10 |
| Demonstration 7b | 5/10 Updated for 6.10 |
| Demonstration 8a | 5/10 Updated for 6.10 |
| Demonstration 8b | 5/10 Updated for 6.10 |
| Demonstration 9 | 5/10 Updated for 6.10 |
| Demonstration 10 | 5/10 Updated for 6.10 |
| Demonstration 11 | 5/10 Updated for 6.10 |
| Demonstration 12 | 5/10 Updated for 6.10 |
| Demonstration 13 | 5/10 Updated for 6.10 |
| Demonstration 14 | 5/10 Updated for 6.10 |
| Demonstration 15 | 5/10 Updated for 6.10 |
| Demonstration 16 | 5/10 Updated for 6.10 |
### Revision Status

<table>
<thead>
<tr>
<th>Workshop</th>
<th>Score</th>
<th>Updated for</th>
</tr>
</thead>
<tbody>
<tr>
<td>Workshop 1</td>
<td>5/10</td>
<td>6.10</td>
</tr>
<tr>
<td>Workshop 2a</td>
<td>5/10</td>
<td>6.10</td>
</tr>
<tr>
<td>Workshop 2b</td>
<td>5/10</td>
<td>6.10</td>
</tr>
<tr>
<td>Workshop 3</td>
<td>5/10</td>
<td>6.10</td>
</tr>
<tr>
<td>Workshop 4a</td>
<td>5/10</td>
<td>6.10</td>
</tr>
<tr>
<td>Workshop 4b</td>
<td>5/10</td>
<td>6.10</td>
</tr>
<tr>
<td>Workshop 5</td>
<td>5/10</td>
<td>6.10</td>
</tr>
<tr>
<td>Workshop 6a</td>
<td>5/10</td>
<td>6.10</td>
</tr>
<tr>
<td>Workshop 6b</td>
<td>5/10</td>
<td>6.10</td>
</tr>
<tr>
<td>Workshop 7a</td>
<td>5/10</td>
<td>6.10</td>
</tr>
<tr>
<td>Workshop 7b</td>
<td>5/10</td>
<td>6.10</td>
</tr>
<tr>
<td>Workshop 7c</td>
<td>5/10</td>
<td>6.10</td>
</tr>
<tr>
<td>Workshop 8</td>
<td>5/10</td>
<td>6.10</td>
</tr>
<tr>
<td>Workshop 9a</td>
<td>5/10</td>
<td>6.10</td>
</tr>
<tr>
<td>Workshop 9b</td>
<td>5/10</td>
<td>6.10</td>
</tr>
<tr>
<td>Workshop 10</td>
<td>5/10</td>
<td>6.10</td>
</tr>
<tr>
<td>Workshop 11</td>
<td>5/10</td>
<td>6.10</td>
</tr>
<tr>
<td>Workshop 12</td>
<td>5/10</td>
<td>6.10</td>
</tr>
<tr>
<td>Workshop 13</td>
<td>5/10</td>
<td>6.10</td>
</tr>
<tr>
<td>Workshop 14a</td>
<td>5/10</td>
<td>6.10</td>
</tr>
<tr>
<td>Workshop 14b</td>
<td>5/10</td>
<td>6.10</td>
</tr>
<tr>
<td>Workshop 15</td>
<td>5/10</td>
<td>6.10</td>
</tr>
<tr>
<td>Workshop 16</td>
<td>5/10</td>
<td>6.10</td>
</tr>
</tbody>
</table>
Overview

• SIMULIA
• What is Abaqus FEA?
• Overview of Abaqus/CAE
• Starting Abaqus/CAE
• Overview of Abaqus/Standard and Abaqus/Explicit
• Abaqus Conventions
• Documentation
• Abaqus Environment Settings
• Abaqus Fetch Utility
• Abaqus/CAE Checklist
• Working with the Model Tree
• Workshop 1: Linear Static Analysis of a Cantilever Beam
Overview

- What are Parts?
- Defining a Part
- Geometry Import and Repair
- Building a Part Using the Part Module Tools
- Workshop 2a: Creating Native Geometry: Pipe Creep Model
- Workshop 2b: Import and Geometry Repair: Intersecting Pipes
Overview

- Details of an Abaqus Input File
- Orphan Mesh Import
- Example
- Workshop 3: Importing and Editing an Orphan Mesh: Pump Model
Overview

- Overview of Abaqus Material Models
- Preliminaries
- Linear Elasticity
- Large Strain Elasticity
- Metal Plasticity
- Material Databases
- Section Properties
- Skins and Stringers (optional)
- Workshop 4a: Material and Section Properties: Pipe Creep Model
- Workshop 4b: Material and Section Properties: Pump Model
Overview

• What is an Assembly?
• Positioning Part Instances
• Patterning
• Boolean Operations
• Sets and Surfaces
• Display Groups
• Workshop 5: Pump Model Assembly
Overview

- Analysis Steps and Procedures
- Output
- Loads and Boundary Conditions
- Workshop 6a: Step Definition and Loads: Pipe Creep Model
- Workshop 6b: Step Definition and Loads: Pump Model
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
- Workshop 7a: Structured Hex Meshing: Pipe Creep Model
- Workshop 7b: Free and Swept Meshing: Pump Model
- Workshop 7c: Meshing of Intersecting Pipes
Overview

- Job Management
- Keywords Editor
- Review of ODB Output
- Results Visualization
  - Basic features
  - Advanced/miscellaneous features
- Workshop 8: Creep of a Pipe Intersection
Overview

- Linear Static Analysis and Multiple Load Cases
- Multiple Load Case Usage
- Examples
- Workshop 9a: Linear Static Analysis of a Cantilever Beam: Multiple Load Cases
- Workshop 9b: Linear Analysis of a Skew Plate
Overview

• Nonlinearity in Structural Mechanics
• Equations of Motion
• Nonlinear Analysis Using Implicit Methods
• Nonlinear Analysis Using Explicit Methods
• Including Nonlinear Effects in an Abaqus Simulation
• Solution Files
• Convergence Issues (Noncontact Related)
• Workshop 10: Nonlinear Analysis of a Skew Plate
Overview

- Multistep Analyses
- Restart Analysis in Abaqus
- Workshop 11: Unloading Analysis of a Skew Plate
Overview

- Introduction
- Tie Constraints
- Rigid Bodies
- Coupling Constraints
- Shell-to-solid Coupling
- Connector Elements
- Fasteners
- Workshop 12: Tie Constraints: Pump Model
Contact

Lecture 13

Overview

• Introduction
• Approaches to Modeling Contact
• Defining General Contact
• Defining Contact Pairs
• Contact Pair Surfaces
• Local Surface Behavior
• Relative Sliding of Points in Contact
• Adjusting Initial Nodal Locations for Contact
• Contact Output
• For More on Contact …
• Workshop 13: Nonlinear Static Analysis of a Pump Assembly
Overview

- What Makes a Problem Dynamic?
- Equations for Dynamic Problems
- Linear Dynamics
- Nonlinear Dynamics
- Comparing Abaqus/Standard and Abaqus/Explicit
- Nonlinear Dynamics Example
- Workshop 14a: Dynamic Analysis of a Skew Plate
- Workshop 14b: Pipe Whip Analysis
Analyzing Highly Nonlinear Quasi-Static Problems
Lecture 15

Overview

• Introduction
• Solution Strategies
• Quasi-Static Simulations Using Explicit Dynamics
• Adaptive Meshing
• Workshop 15: Single Pass Rolling of a Thick Plate
Overview

- Introduction
- Steady-State Heat Transfer
- Transient Heat Transfer
- Thermal Interfaces
- Thermal-Stress Analysis
- Workshop 16: Thermal-Stress Analysis of Intersecting Pipes
Overview

• What are Finite Elements?
• Objectives of Finite Element Analysis
• Components of an Analysis Model
Overview

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

Appendix 3

Overview

• Contact Issues Specific to Abaqus/Standard
• Contact Issues Specific to Abaqus/Explicit