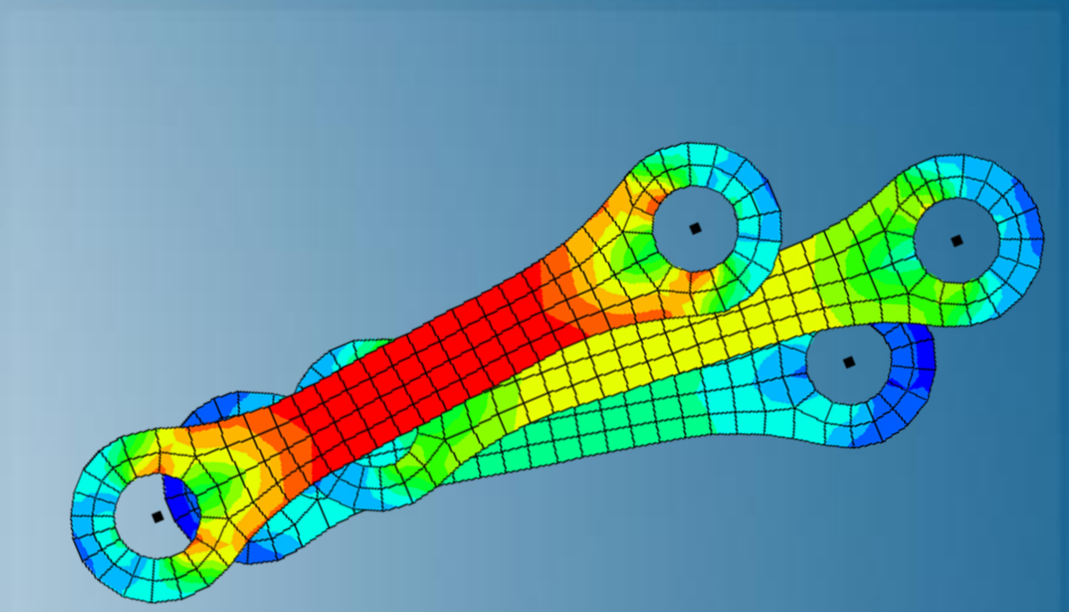
Substructures and Submodeling with Abaqus

2017



3DEXPERIENCE

About this Course

Course objectives

Upon completion of this course you will be able to:

- Understand the difference between substructuring and submodeling
- Build, translate, rotate and reflect substructures
- Build preloads into substructures
- Design meshes for submodel analysis
- Perform solid-to-solid, shell-to-shell, and shell-to-solid submodeling

Targeted audience

Simulation Analysts

Prerequisites

This course is recommended for engineers with experience using Abaqus

2 days

Day 1

- Lecture 1 — Introduction to Substructures
- Lecture 2 — Using Static Substructuring in Abaqus
- Lecture 3 — Linear Perturbations about a Preloaded State
- Lecture 4 — Dynamic Substructuring
- Lecture 5 — Substructure Output

- Lecture 6 — Substructuring Examples
- Workshop 1a — Substructures: Plane Frame Analysis
- Workshop 1b — Substructures: Surface Mount Analysis

- Lecture 7 — Using substructures with Abaqus/Explicit
- Workshop 2 — Substructures: Beam Impact (*optional*)

Day 2

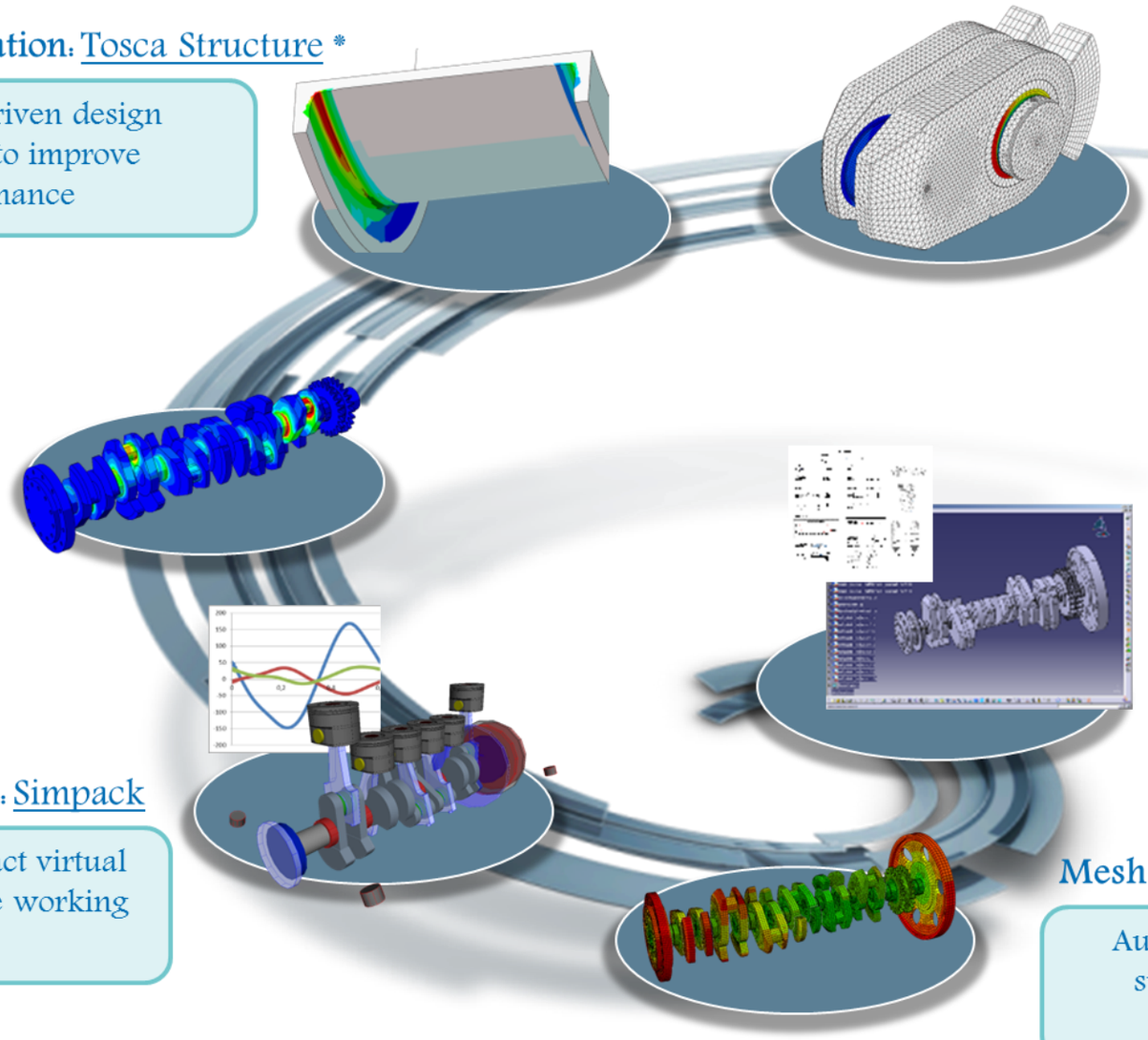
- Lecture 8 — Introduction to Submodeling
- Lecture 9 — Submodeling in Abaqus
- Lecture 10 — Abaqus Usage and Examples (Part 1)
- Workshop 3 — Submodeling: Pressure Vessel Nozzle Analysis
- Lecture 11 — Abaqus Usage and Examples (Part 2)
- Workshop 4 — Submodeling: Ceramic-Metal Braze Joint
- Lecture 12 — Submodeling Practices
- Workshop 5 — Submodeling: Composite Tube Joint
- Lecture 13 — Limitations of Submodeling

Additional Material

- Appendix 1 Theory of Substructures

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 *

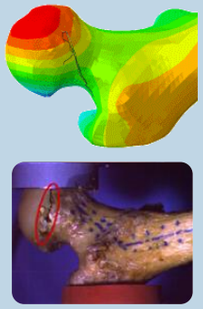
Automated mesh calibration: sufficient mesh quality for accurate results

* Included in extended licensing pool

SIMULIA's Power of the Portfolio

Abaqus

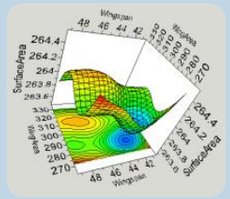
- Routine and Advanced Simulation
- Linear and Nonlinear, Static and Dynamic
- Thermal, Electrical, Acoustics
- Extended Physics through Co-simulation
- Model Preparation and Visualization



Realistic Human Simulation
High Speed Crash & Impact
Noise & Vibration

Isight

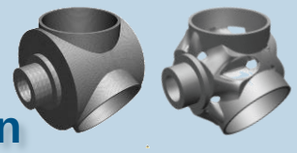
- Process Integration
- Design Optimization
- Parametric Optimization
- Six Sigma and Design of Experiments



Material Calibration
Workflow Automation
Design Exploration

Tosca

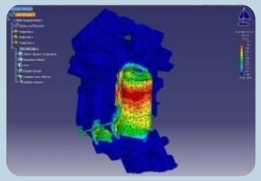
- Non-Parametric Optimization
- Structural and Fluid Flow Optimization
- Topology, Sizing, Shape, Bead Optimization



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

fe-safe

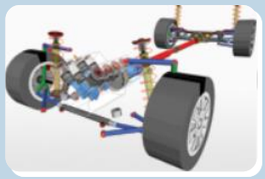
- Durability Simulation
- Low Cycle and High Cycle Fatigue
- Weld, High Temperature, Non-metallics



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

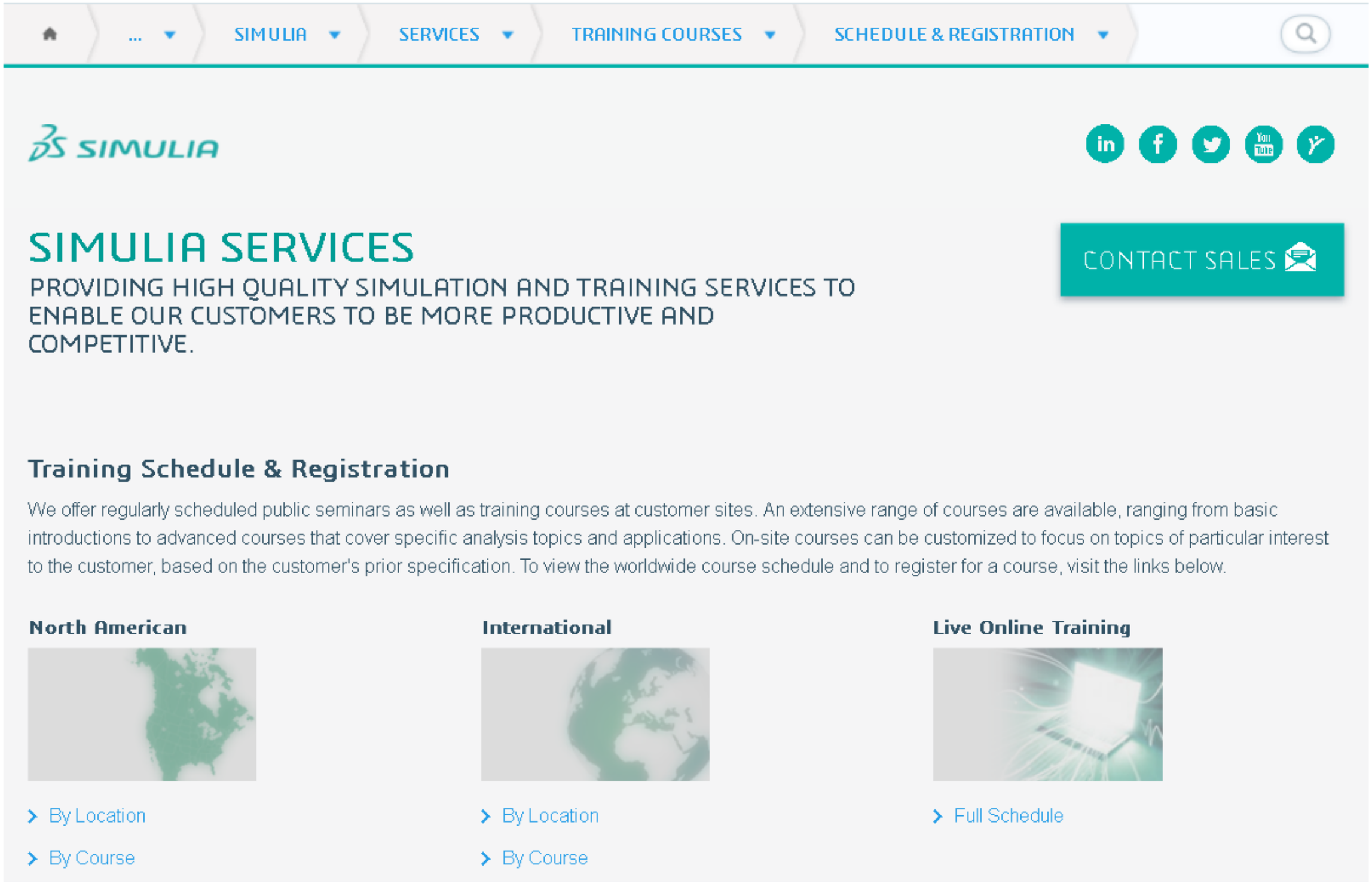
Discover new ways to explore how to leverage realistic simulation to drive product innovation. Join the thousands of Abaqus and Isight users who are already gaining valuable knowledge from the SIMULIA Learning Community.

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	11/16	Updated for Abaqus 2017
Lecture 2	11/16	Updated for Abaqus 2017
Lecture 3	11/16	Updated for Abaqus 2017
Lecture 4	11/16	Updated for Abaqus 2017
Lecture 5	11/16	Updated for Abaqus 2017
Lecture 6	11/16	Updated for Abaqus 2017
Lecture 7	11/16	Updated for Abaqus 2017
Lecture 8	11/16	Updated for Abaqus 2017
Lecture 9	11/16	Updated for Abaqus 2017
Lecture 10	11/16	Updated for Abaqus 2017
Lecture 11	11/16	Updated for Abaqus 2017
Lecture 12	11/16	Updated for Abaqus 2017
Lecture 13	11/16	Updated for Abaqus 2017
Appendix 1	11/16	Updated for Abaqus 2017

Workshop 1a	11/16	Updated for Abaqus 2017
Workshop 1b	11/16	Updated for Abaqus 2017
Workshop 2	11/16	Updated for Abaqus 2017
Workshop 3	11/16	Updated for Abaqus 2017
Workshop 4	11/16	Updated for Abaqus 2017
Workshop 5	11/16	Updated for Abaqus 2017

Lesson 1: Introduction to Substructures

Lesson content:

- Why Substructuring?
- Static Substructuring
- Advantages of Substructuring
- Procedures Supporting Substructures

30 minutes

Lesson 2: Using Static Substructuring in Abaqus

Lesson content:

- The Basics
- Substructure Generation
- Substructure Usage: Abaqus/CAE
- Substructure Usage: Keywords
- Substructure Load Cases
- Substructure Gravity Loading
- Kinematic Constraints in Substructures
- Flexible Body Dynamics
- Limitations

1 hour

Lesson 3: Linear Perturbations about a Preloaded State

Lesson content:

- Introduction
- Substructure Tangent Stiffness Calculation
- Response Quantities
- Effect of Preloads at the Usage Level
- Preloading Syntax
- Preloading Example: Rotating Structure

30 minutes

Lesson 4: Dynamic Substructuring

Lesson content:

- Guyan Reduction
- Dynamic Mode Addition
- Damping with Substructures

45 minutes

Lesson 5: Substructure Output

Lesson content:

- Introduction
- Visualizing Substructure Results
- Output of Eliminated Degrees of Freedom
- Output of Substructure Matrices
- Substructure Library Utilities

45 minutes

Lesson 6: Substructuring Examples

Lesson content:

- Cyclic Symmetry
- Multilevel Substructuring
- Workshop Preliminaries
- Workshop 1a: Substructures: Plane Frame Analysis (IA)
- Workshop 1a: Substructures: Plane Frame Analysis (KW)
- Workshop 1b: Substructures: Surface Mount Analysis (IA)
- Workshop 1b: Substructures: Surface Mount Analysis (KW)

Both interactive (IA) and keywords (KW) versions of the workshop are provided. Complete only one.

3 hours

Lesson 7: Using Substructures with Abaqus/Explicit

Lesson content:

- Introduction
- Examples
- General Concepts
- Keyword Interface
- Interactive Interface
- Postprocessing
- Technology Notes
- Workshop 2: Substructures: Beam Impact (IA)
- Workshop 2: Substructures: Beam Impact (KW)

Both interactive (IA) and keywords (KW) versions of the workshop are provided. Complete only one.

2 hours

Lesson 8: Introduction to Submodeling

Lesson content:

- Concept of Submodeling
- Motivation for Submodeling

15 minutes

Lesson 9: Submodeling in Abaqus

Lesson content:

- Fundamental Assumptions
- Submodeling Techniques
- Node-based Implementation
- Surface-based Implementation

30 minutes

Lesson 10: Abaqus Usage and Examples (Part 1)

Lesson content:

- Terminology
- Transfer of Data
- Prescribed Values
- Submodeling Workflow
- Surface-Based Submodel Boundaries
- Example: Conical Crack in a Half Space
- Example: Pressure Vessel
- Workshop 3: Submodeling: Pressure Vessel Nozzle Analysis (IA)
- Workshop 3: Submodeling: Pressure Vessel Nozzle Analysis (KW)

Both interactive (IA) and keywords (KW) versions of the workshop are provided. Complete only one.

2.5 hours

Lesson 11: Abaqus Usage and Examples (Part 2)

Lesson content:

- Node-Based Submodel Boundaries
- Example: Stacked Sheet Metal Assembly
- Example: Large Displacement Analysis
- Tolerances at the Submodel Boundary
- Shell-to-Solid Submodeling
- Example: Shell-to-Solid Submodel of a Pipe Joint
- Workshop 4: Submodeling: Ceramic-Metal Braze Joint (IA)
- Workshop 4: Submodeling: Ceramic-Metal Braze Joint (KW)

Both interactive (IA) and keywords (KW) versions of the workshop are provided. Complete only one.

2 hours

Lesson 12: Submodeling Practices

Lesson content:

- Perturbation Analysis
- Changing Procedures
- The Frequency Domain
- Submodeling and Thermal Stress Analysis
- Example: Thermal Strain in a Bar
- Submodeling in Dynamic Procedures
- Example: Speaker Diaphragm
- Workshop 5: Submodeling: Composite Tube Joint (IA)
- Workshop 5: Submodeling: Composite Tube Joint (KW)

Both interactive (IA) and keywords (KW) versions of the workshop are provided. Complete only one.

1 hour

Lesson 13: Limitations of Submodeling

Lesson content:

- Elements
- Procedures
- Shell-to-Solid

15 minutes

Appendix 1: Theory of Substructures

Appendix content:

- Static Substructuring
- Guyan Reduction
- Restrained Mode Addition

15 minutes