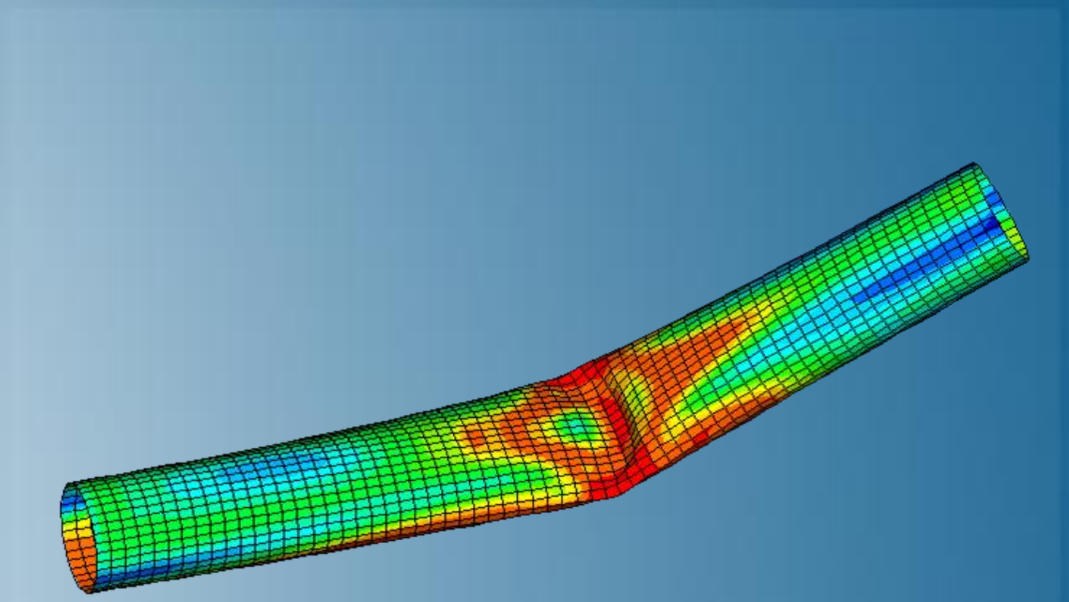
Buckling, Postbuckling, and Collapse Analysis with Abaqus

Abaqus 2018



3DEXPERIENCE®

About this Course

Course objectives

Upon completion of this course you will be able to:

- Perform linear eigenvalue buckling analysis
- Perform postbuckling analysis using the regular and damped static solution procedures
- Perform postbuckling analysis using the modified Riks method
- Perform postbuckling analysis using dynamics solution procedures

Targeted audience

Simulation Analysts

Prerequisites

This course is recommended for engineers with experience using Abaqus

2 days

Day 1

- Lecture 1 — Basic Concepts and Overview
- Lecture 2 — Linear and Nonlinear FEA with Abaqus
- Lecture 3 — Eigenvalue Buckling Analysis
  - Workshop 1 — Elastic Buckling of a Stiffened Cylindrical Shell
  - Workshop 2 — Eigenvalue Buckling of a Ring (optional)
- Lecture 4 — Regular Static Solution Procedure
  - Workshop 3 — Nonlinear Buckling of a Stiffened Cylindrical Shell

Day 2

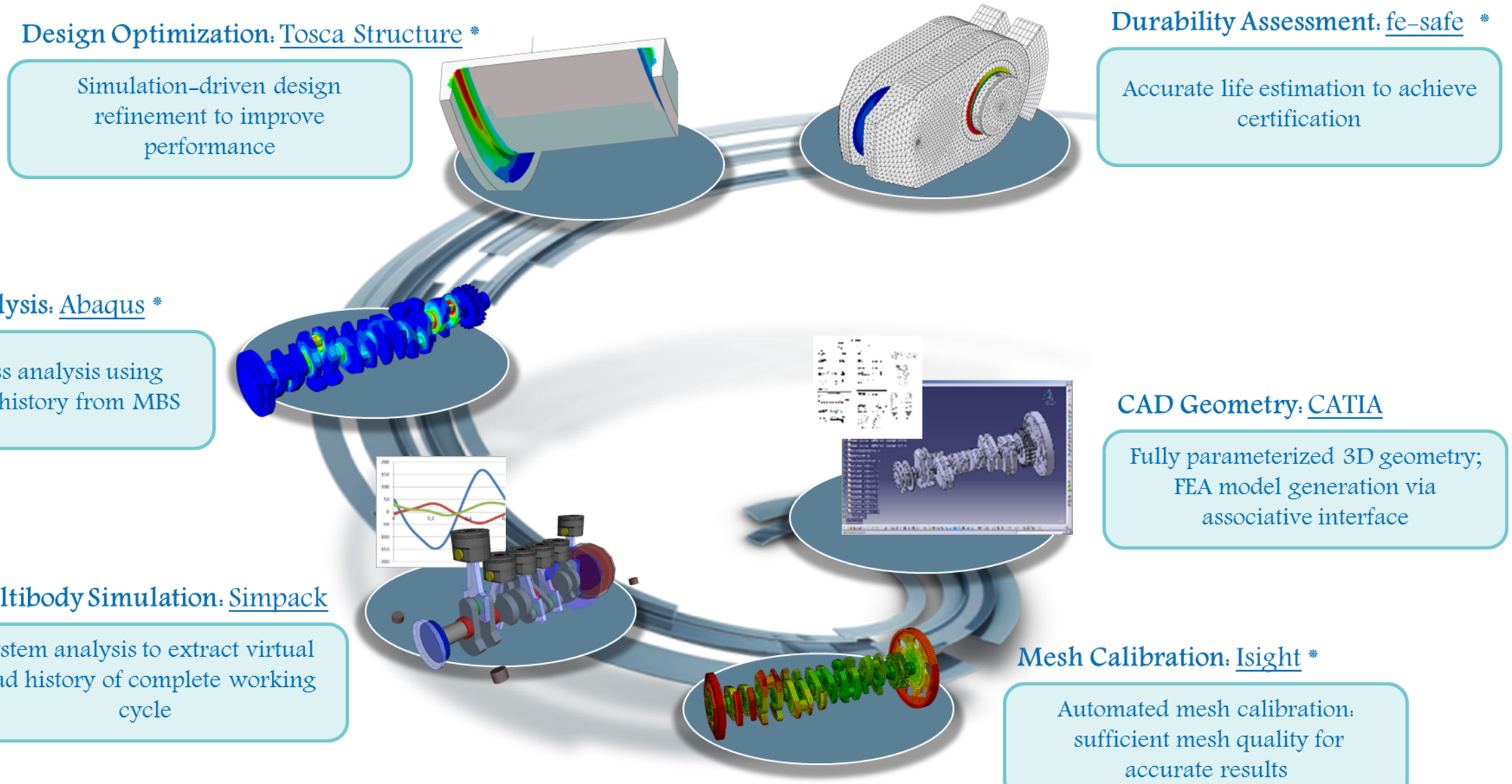
- Lecture 5 Damped Static Solution Procedure
  - Workshop 3 Nonlinear Buckling of a Stiffened Cylindrical Shell (continued)
  - Workshop 4 Static Buckling Analysis of a Circular Arch
- Lecture 6 Modified Riks Static Solution Procedure
  - Workshop 4 Static Buckling Analysis of a Circular Arch (continued)
- Lecture 7 Dynamic Analysis Solution Procedures
  - Workshop 4 Static Buckling Analysis of a Circular Arch (continued)
  - Workshop 5 Tube Crush Dynamic Analysis
- Lecture 8 Putting It All Together…
  - Workshop 6 Lee's Frame Buckling Problem
  - Workshop 7 Buckling and Postbuckling of a Crane Structure (optional)
  - Workshop 8 Buckling and Postbuckling of a Stiffened Panel (optional)

Additional Material

- Appendix 1 Geometrically Nonlinear Analysis
- Appendix 2 Dashpots

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 *

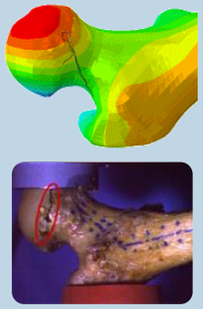
Automated mesh calibration: sufficient mesh quality for accurate results

* Included in extended licensing pool

SIMULIA's Power of the Portfolio

Abaqus

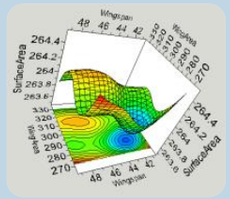
- Routine and Advanced Simulation
- Linear and Nonlinear, Static and Dynamic
- Thermal, Electrical, Acoustics
- Extended Physics through Co-simulation
- Model Preparation and Visualization



Realistic Human Simulation
High Speed Crash & Impact
Noise & Vibration

Isight

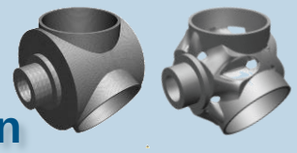
- Process Integration
- Design Optimization
- Parametric Optimization
- Six Sigma and Design of Experiments



Material Calibration
Workflow Automation
Design Exploration

Tosca

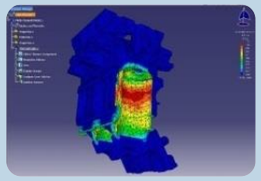
- Non-Parametric Optimization
- Structural and Fluid Flow Optimization
- Topology, Sizing, Shape, Bead Optimization



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

fe-safe

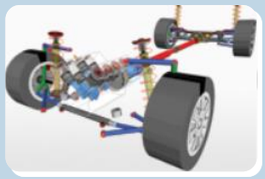
- Durability Simulation
- Low Cycle and High Cycle Fatigue
- Weld, High Temperature, Non-metallics



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

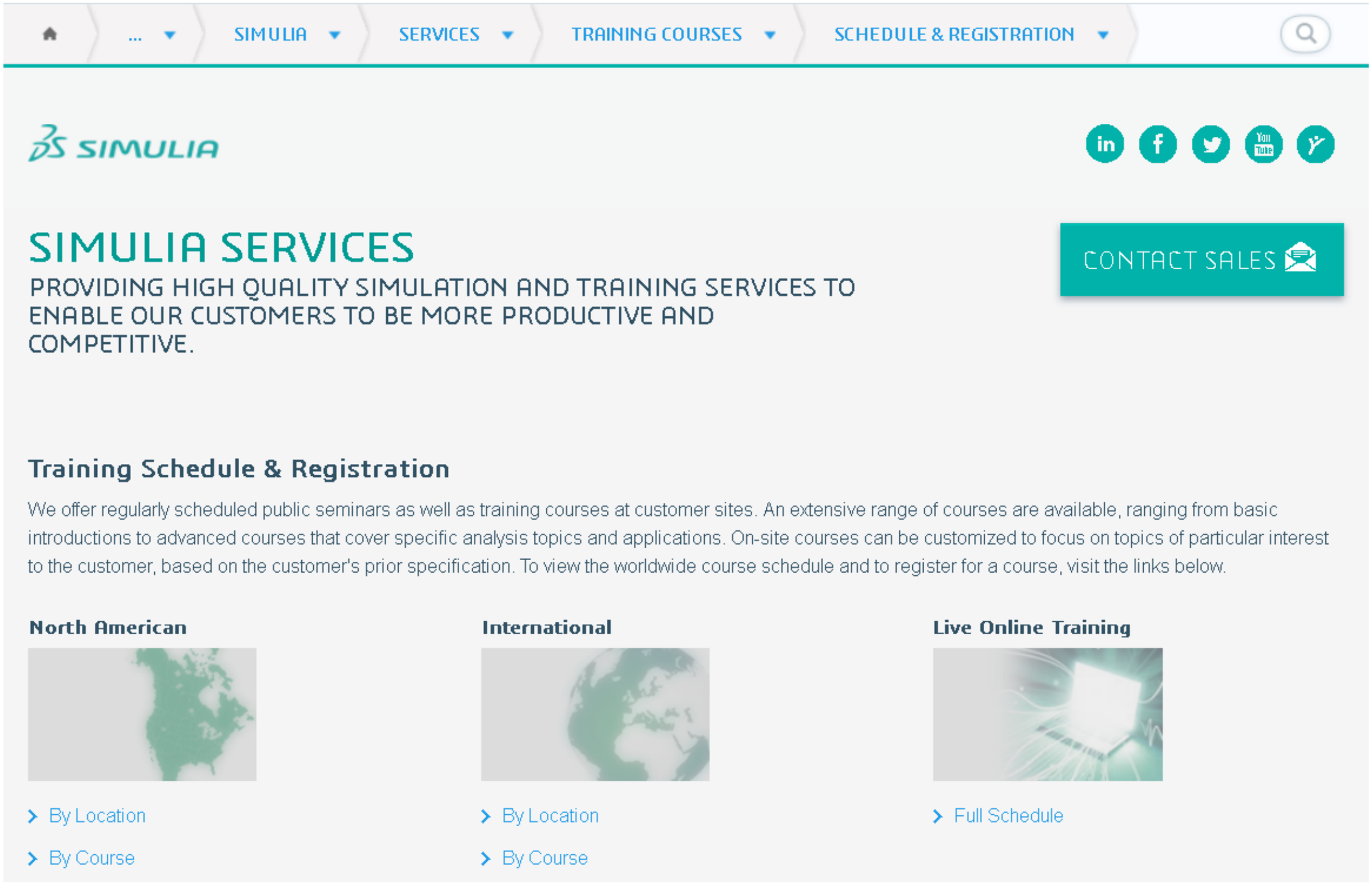
Discover new ways to explore how to leverage realistic simulation to drive product innovation. Join the thousands of Abaqus and Isight users who are already gaining valuable knowledge from the SIMULIA Learning Community.

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	11/17	Updated for Abaqus 2018
Lecture 2	11/17	Updated for Abaqus 2018
Lecture 3	11/17	Updated for Abaqus 2018
Lecture 4	11/17	Updated for Abaqus 2018
Lecture 5	11/17	Updated for Abaqus 2018
Lecture 6	11/17	Updated for Abaqus 2018
Lecture 7	11/17	Updated for Abaqus 2018
Lecture 8	11/17	Updated for Abaqus 2018
Appendix 1	11/17	Updated for Abaqus 2018
Appendix 2	11/17	Updated for Abaqus 2018
Workshop 1	11/17	Updated for Abaqus 2018
Workshop 2	11/17	Updated for Abaqus 2018
Workshop 3	11/17	Updated for Abaqus 2018
Workshop 4	11/17	Updated for Abaqus 2018
Workshop 5	11/17	Updated for Abaqus 2018
Workshop 6	11/17	Updated for Abaqus 2018
Workshop 7	11/17	Updated for Abaqus 2018
Workshop 8	11/17	Updated for Abaqus 2018

Lesson 1: Basic Concepts and Overview

Lesson content:

- Introduction
- Revisiting Classical Stability Problems
- Solution Techniques
- Example: Lee’s Frame Buckling Problem
- Summary

1 hour

Lesson 2: Linear and Nonlinear FEA with Abaqus

Lesson content:

- Basic Equations in Finite Element Analysis
- Linearization
- Nonlinear Problems in Mechanics
- General and Perturbation Procedures
- Including Nonlinear Effects in an Abaqus Simulation
- Summary

1 hour

Lesson 3: Eigenvalue Buckling Analysis

Lesson content:

- Introduction
- Eigenvalue Problem Formulation
- Abaqus Usage
- Example: Buckling of a Thin Cylindrical Shell
- Closely Spaced Eigenvalues
- Boundary Conditions and Symmetry in Buckling Analyses
- Concluding Remarks
- Workshop Preliminaries
- Workshop 1: Elastic Buckling of a Stiffened Cylindrical Shell (IA)
- Workshop 1: Elastic Buckling of a Stiffened Cylindrical Shell (KW)
- Workshop 2: Eigenvalue Buckling of a Ring (IA)
- Workshop 2: Eigenvalue Buckling of a Ring (KW)

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

2 hours

Lesson 4: Regular Static Solution Procedure

Lesson content:

- Introduction
- Introducing Imperfections for Postbuckling Simulations
- Solving Nonlinear Problems with Implicit Techniques
- Solution Control
- Automatic Time Incrementation
- Diagnostic Information
- Limitations of Regular Static Procedure
- Concluding Remarks
- Workshop 3: Nonlinear Buckling of a Stiffened Cylindrical Shell (IA)
- Workshop 3: Nonlinear Buckling of a Stiffened Cylindrical Shell (KW)

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

2.5 hours

Lesson 5: Damped Static Solution Procedure

Lesson content:

- Introduction
- Damping in Static Analyses
- Automatic Stabilization
- Automatic Stabilization Examples
- Postbuckling and Loss of Contact
- Concluding Remarks
- Workshop 3: Nonlinear Buckling of a Stiffened Cylindrical Shell (IA, cont'd)
- Workshop 3: Nonlinear Buckling of a Stiffened Cylindrical Shell (KW, cont'd)
- Workshop 4: Static Buckling Analysis of a Circular Arch (IA)
- Workshop 4: Static Buckling Analysis of a Circular Arch (KW)

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

2 hours

Lesson 6: Modified Riks Static Solution Procedure

Lesson content:

- Introduction
- Abaqus Implementation
- Abaqus Usage
- Snap-Through Problems
- Postbuckling Problems
- Postbuckling Examples
- Usage Tips
- Limitations
- Concluding Remarks
- Workshop 4 (continued): Static Buckling Analysis of a Circular Arch (IA)
- Workshop 4 (continued): Static Buckling Analysis of a Circular Arch (KW)

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

1.5 hours

Lesson 7: Dynamic Analysis Solution Procedures

Lesson content:

- Overview
- What Makes a Problem Dynamic?
- Equations for Dynamic Problems
- Nonlinear Dynamics
- Comparing Abaqus/Standard and Abaqus/Explicit
- Workshop 4 (continued): Static Buckling Analysis of a Circular Arch (IA)
- Workshop 4 (continued): Static Buckling Analysis of a Circular Arch (KW)
- Analyzing Highly Nonlinear Quasi-Static Problems
- Quasi-Static Simulations Using Explicit Dynamics
- Example: Dynamic Tube Collapse
- Concluding Remarks
- Workshop 5: Tube Crush Dynamic Analysis (IA)
- Workshop 5: Tube Crush Dynamic Analysis (KW)

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

2.5 hours

Lesson 8: Putting It All Together…

Lesson content:

- Buckling Analysis Selection Guide
- Weatherseal Example
- References for Further Study
- Workshop 6: Lee's Frame Buckling Problem (IA)
- Workshop 6: Lee's Frame Buckling Problem (KW)
- Workshop 7: Buckling and Postbuckling of a Crane Structure (IA)
- Workshop 7: Buckling and Postbuckling of a Crane Structure (KW)
- Workshop 8: Buckling and Postbuckling of a Stiffened Panel (IA)
- Workshop 8: Buckling and Postbuckling of a Stiffened Panel (KW)

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

1.5 hours

Appendix 1: Geometrically Nonlinear Analysis

Appendix content:

- Introduction
- Equilibrium and Virtual Work
- Deformation and Strain
- Large Rotations
- Follower Forces

30 minutes

Appendix 2: Dashpots

Appendix content:

- Dashpots

15 minutes