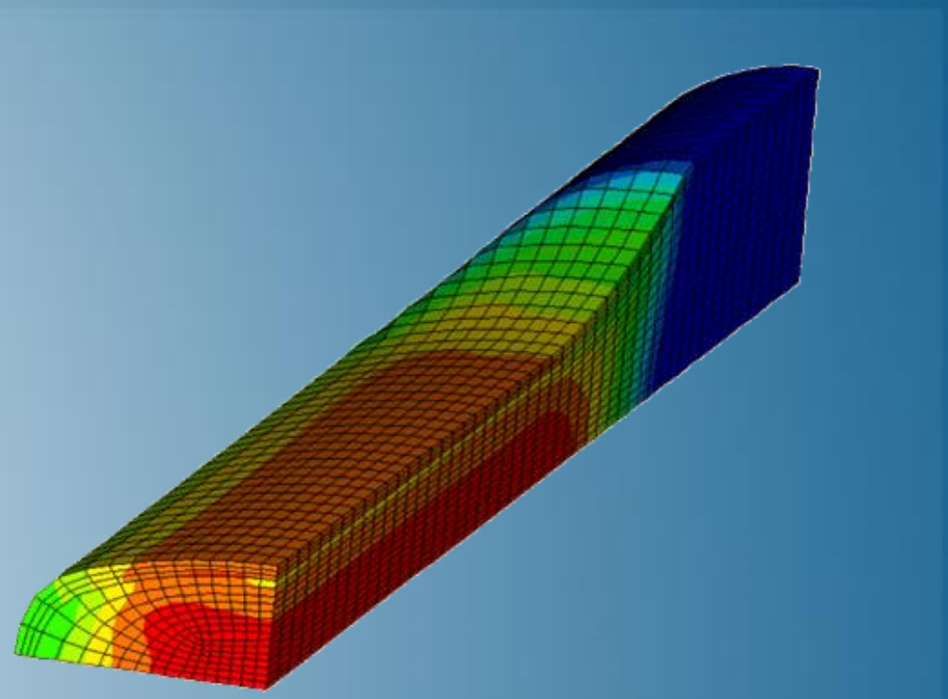
Metal Forming with Abaqus

Abaqus 2019



3DEXPERIENCE®

About this Course

Course objectives

In this course you will learn practical modeling skills and techniques for:

- Stamping
- Hydroforming
- Punch stretching
- Forging
- Rolling
- Drawing
- Superplastic forming

Targeted audience

This course is recommended for engineers with experience using Abaqus

Prerequisites

None

3 days

Day 1

- Lecture 1 Introduction
- Lecture 2 Solution Procedures in Abaqus
- Lecture 3 Contact
  - Workshop 1 Bulk Forming of a Cup
- Lecture 4 Elements
- Lecture 5 Materials

Day 2

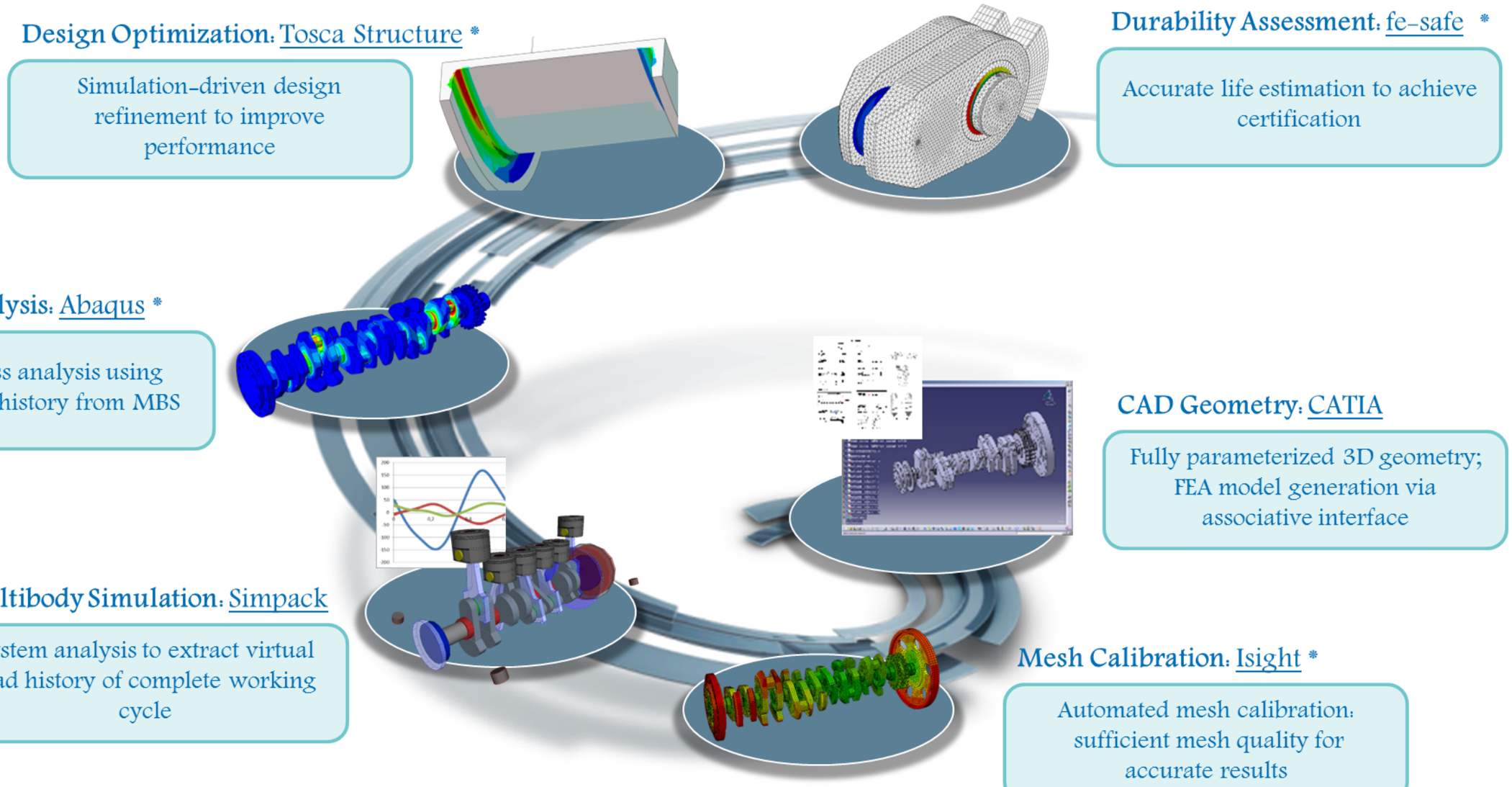
- Lecture 6 Adaptive Meshing
- Lecture 7 Modeling Quasi-Static Processes Using Abaqus/Explicit
  - Workshop 2 Rolling of a Thick Plate
- Lecture 8 Transferring Results between Abaqus Analyses
  - Workshop 3 Production of an Angle Bracket
- Lecture 9 Model Change
- Lecture 10 Thermal Effects

Day 3 (Selected topics as time permits)

- Lecture 11 Rolling Analysis
- Lecture 12 Multi-Pass Rolling
- Lecture 13 Drawbead Modeling
- Lecture 14 Hydroforming
- Lecture 15 Superelastic Forming Analysis with Abaqus

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 *

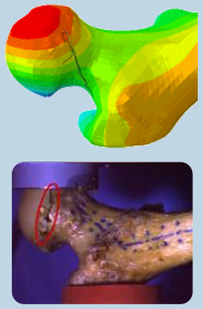
Automated mesh calibration: sufficient mesh quality for accurate results

* Included in extended licensing pool

SIMULIA's Power of the Portfolio

Abaqus

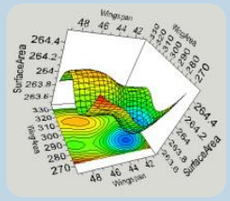
- Routine and Advanced Simulation
- Linear and Nonlinear, Static and Dynamic
- Thermal, Electrical, Acoustics
- Extended Physics through Co-simulation
- Model Preparation and Visualization



Realistic Human Simulation
High Speed Crash & Impact
Noise & Vibration

Isight

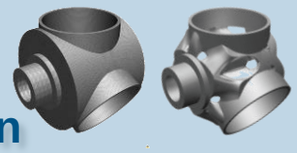
- Process Integration
- Design Optimization
- Parametric Optimization
- Six Sigma and Design of Experiments



Material Calibration
Workflow Automation
Design Exploration

Tosca

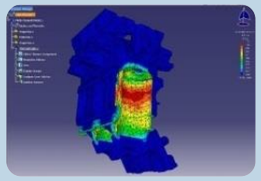
- Non-Parametric Optimization
- Structural and Fluid Flow Optimization
- Topology, Sizing, Shape, Bead Optimization



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

fe-safe

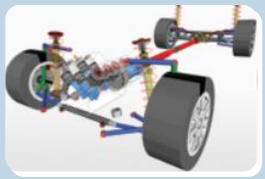
- Durability Simulation
- Low Cycle and High Cycle Fatigue
- Weld, High Temperature, Non-metallics



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

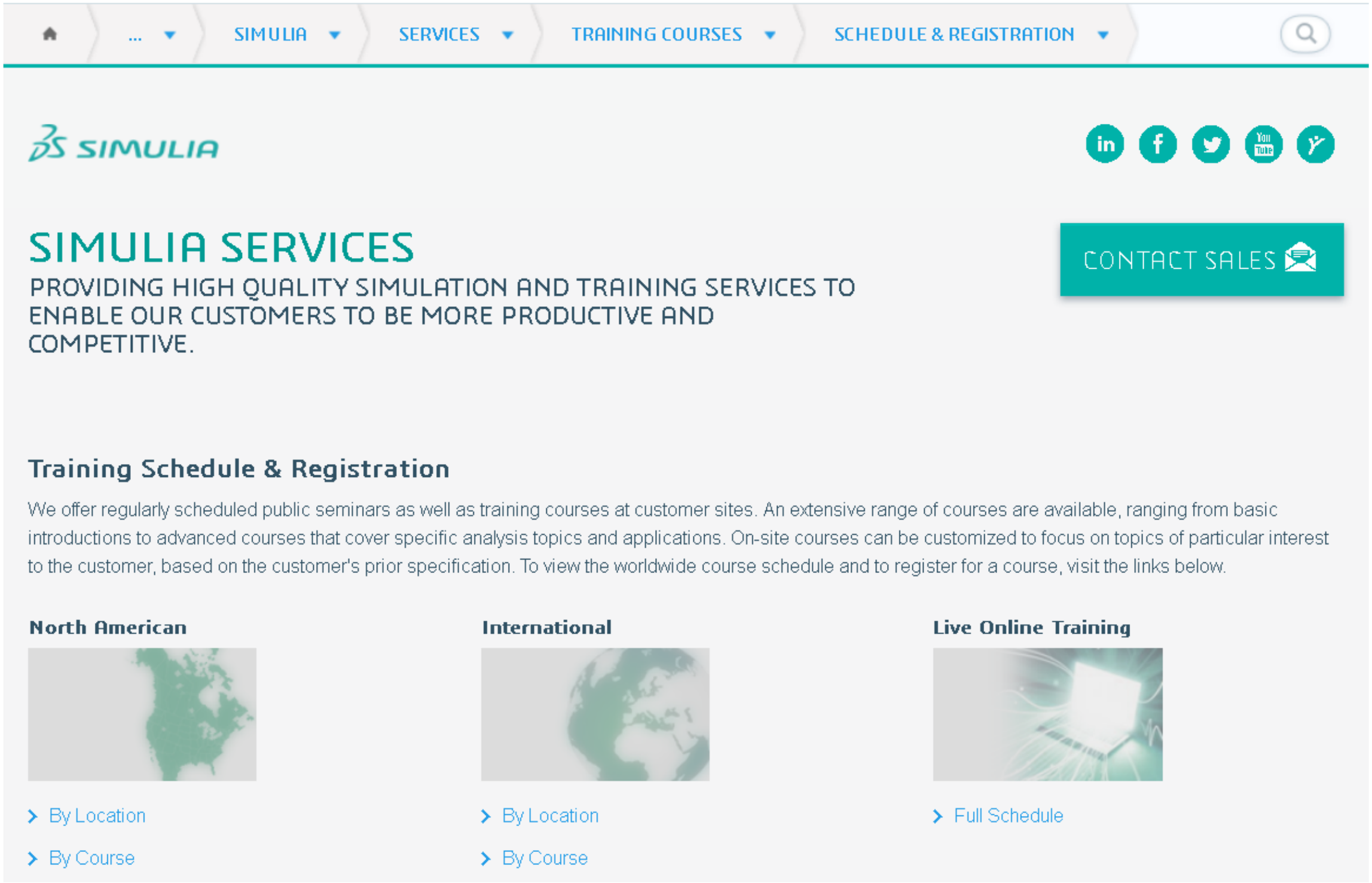
Discover new ways to explore how to leverage realistic simulation to drive product innovation. Join the thousands of Abaqus and Isight users who are already gaining valuable knowledge from the SIMULIA Learning Community.

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	11/18	Updated for Abaqus 2019
Lecture 2	11/18	Updated for Abaqus 2019
Lecture 3	11/18	Updated for Abaqus 2019
Lecture 4	11/18	Updated for Abaqus 2019
Lecture 5	11/18	Updated for Abaqus 2019
Lecture 6	11/18	Updated for Abaqus 2019
Lecture 7	11/18	Updated for Abaqus 2019
Lecture 8	11/18	Updated for Abaqus 2019
Lecture 9	11/18	Updated for Abaqus 2019
Lecture 10	11/18	Updated for Abaqus 2019
Lecture 11	11/18	Updated for Abaqus 2019
Lecture 12	11/18	Updated for Abaqus 2019
Lecture 13	11/18	Updated for Abaqus 2019
Lecture 14	11/18	Updated for Abaqus 2019
Lecture 15	11/18	Updated for Abaqus 2019
Workshop 1	11/18	Updated for Abaqus 2019
Workshop 2	11/18	Updated for Abaqus 2019
Workshop 3	11/18	Updated for Abaqus 2019

Lesson 1: Introduction

Lesson content:

- Introduction
- Forming Processes
- Motivation Behind Metal Forming Simulation

45 minutes

Lesson 2: Solution Procedures with Abaqus

Lesson content:

- Introduction
- Equilibrium
- Implicit Solution of Static Equilibrium
- Explicit Solution of Dynamic Equilibrium
- Implicit and Explicit Procedures for Metal Forming

45 minutes

Lesson 3: Contact

Lesson content:

- Introduction to Modeling Contact
- Defining General Contact
- General Contact Output
- Limitations of General Contact
- Contact Pairs
- Contact Pair Output
- Contact Constraint Algorithm
- Friction
- Contact Modeling Tips
- Rigid Bodies in Abaqus
- Workshop Preliminaries
- Workshop 1: Bulk Forming of a Cup (IA)
- Workshop 1: Bulk Forming of a Cup (KW)

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

3 hours

Lesson 4: Elements

Lesson content:

- Introduction
- Continuum Elements
- Structural Elements
- Special-Purpose Elements
- Element Selection in Scoping Studies
- Hourglassing
- Secondary-order Accuracy

1.5 hours

Lesson 5: Materials

Lesson content:

- Introduction
- Elasticity
- Mises Plasticity
- Anisotropic (Hill's) Plasticity
- Gurson Model
- Rate Dependence
- Annealing
- Forming Limit Diagrams

1.5 hours

Lesson 6: Adaptive Meshing

Lesson content:

- Introduction to Adaptive Meshing
- Arbitrary Lagrangian-Eulerian (ALE) Method
- Lagrangian Adaptive Mesh Domains
- Eulerian Adaptive Mesh Domains for Steady-state Analyses
- Additional Features of Adaptive Meshing
- Adaptive Meshing Output and Diagnostics
- Summary

1.5 hours

Lesson 7: Quasi-Static Analyses

Lesson content:

- Introduction
- Quasi-Static Simulations Using Explicit Dynamics
- Loading Rates
- Energy Balance in Quasi-Static Analyses
- Mass Scaling
- Viscous Pressure
- Summary
- Workshop 2: Rolling of a Thick Plate (IA)
- Workshop 2: Rolling of a Thick Plate (KW)

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

2.5 hours

Lesson 8: Transferring Results between Abaqus Analyses

Lesson content:

- Introduction
- Import from Abaqus/Explicit to Abaqus/Standard
- Import from Abaqus/Standard to Abaqus/Explicit
- Import from Abaqus/Explicit to Abaqus/Explicit
- Additional Import Modeling Issues
- Limitations
- Demonstration
- Workshop 3: Production of an Angle Bracket (IA)
- Workshop 3: Production of an Angle Bracket (KW)

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

2 hours

Lesson 9: Model Change

Lesson content:

- Introduction
- Multistage Forming Processes
- Element Removal in Abaqus/Standard

30 minutes

Lesson 10: Thermal Effects

Lesson content:

- Introduction
- Fully Coupled Analysis
- Adiabatic Analysis
- Rigid Bodies in Thermal-Stress Analysis

45 minutes

Lesson 11: Rolling Analysis

Lesson content:

- Introduction
- Hot Rolling with Abaqus/Explicit
- Single Pass Simulation
- Steady-State Analysis
- Transient Analysis

45 minutes

Lesson 12: Multi-Pass Rolling Analysis

Lesson content:

- Multi-Pass Simulation

30 minutes

Lesson 13: Drawbead Modeling

Lesson content:

- Introduction
- Drawbead Restraint Forces
- Nonlinear Springs
- Point Masses with Node-Based Contact
- Example: Forming of a Fender

45 minutes

Lesson 14: Hydroforming

Lesson content:

- Introduction
- Hydroforming Processes
- Hydroforming Example

30 minutes

Lesson 15: Superplastic Forming Analysis with Abaqus

Lesson content:

- Introduction
- Superplastic Forming Modeling
- Superplastic Forming Example

30 minutes