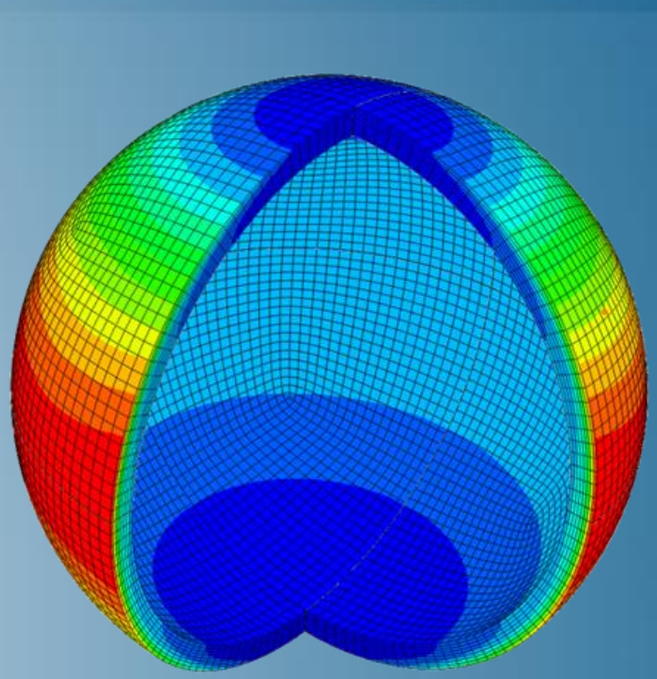
# Electromagnetic Analysis with Abaqus

## Abaqus 2018




3DEXPERIENCE®

# About this Course

## Course objectives

Upon completion of this course you will be able to:

- Set up and create electromagnetic models with Abaqus
- Perform low frequency eddy current analyses with Abaqus
- Perform transient eddy current analyses with Abaqus
- Perform magnetostatic analyses with Abaqus

## Targeted audience

Simulation Analysts

## Prerequisites

This course is recommended for engineers with experience using Abaqus

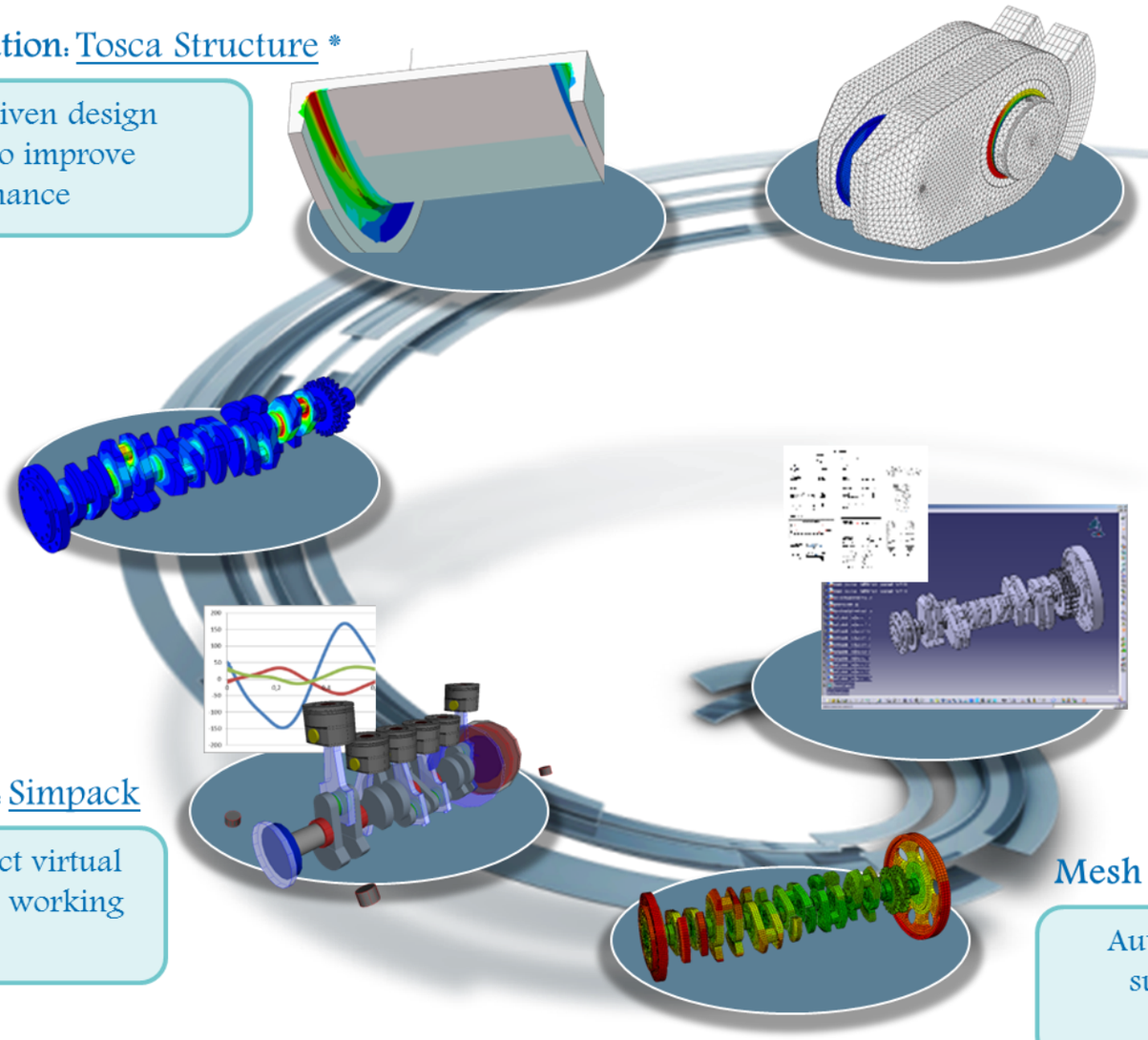
**1 day**

# Day 1

- Lecture 1 — Introduction to Computational Electromagnetics
- Lecture 2 — Geometry, Material Properties, Elements and Meshing
  - Workshop 1 — Heating of a Rod: Problem setup
  - Workshop 2 — Sphere in a Magnetic Field: Problem setup
- Lecture 3 — Loads and Boundary Conditions
- Lecture 4 — Output and Transfer of Results
  - Workshop 1 (cont'd) — Heating of a Rod: Thermal Response
  - Workshop 2 (cont'd) — Sphere in a Magnetic Field: Electromagnetic Response
  - Workshop 3 — Magnetostatic Analysis of a Solenoid Valve
  - Workshop 4 — Magnetic Pulse Forming of a Metallic Tube

# 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 ***

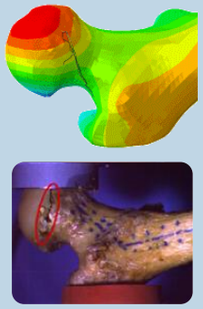
Automated mesh calibration: sufficient mesh quality for accurate results

* Included in extended licensing pool

# SIMULIA's Power of the Portfolio

**Abaqus**

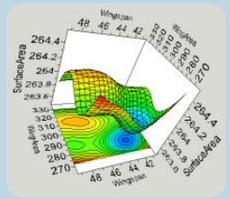
- Routine and Advanced Simulation
- Linear and Nonlinear, Static and Dynamic
- Thermal, Electrical, Acoustics
- Extended Physics through Co-simulation
- Model Preparation and Visualization



**Realistic Human Simulation**
**High Speed Crash & Impact**
**Noise & Vibration**

**Isight**

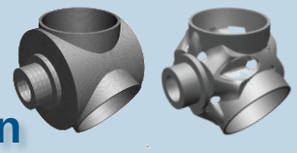
- Process Integration
- Design Optimization
- Parametric Optimization
- Six Sigma and Design of Experiments



**Material Calibration**
**Workflow Automation**
**Design Exploration**

**Tosca**

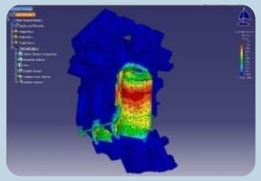
- Non-Parametric Optimization
- Structural and Fluid Flow Optimization
- Topology, Sizing, Shape, Bead Optimization



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

**fe-safe**

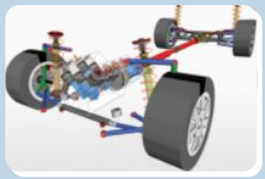
- Durability Simulation
- Low Cycle and High Cycle Fatigue
- Weld, High Temperature, Non-metallics



**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**

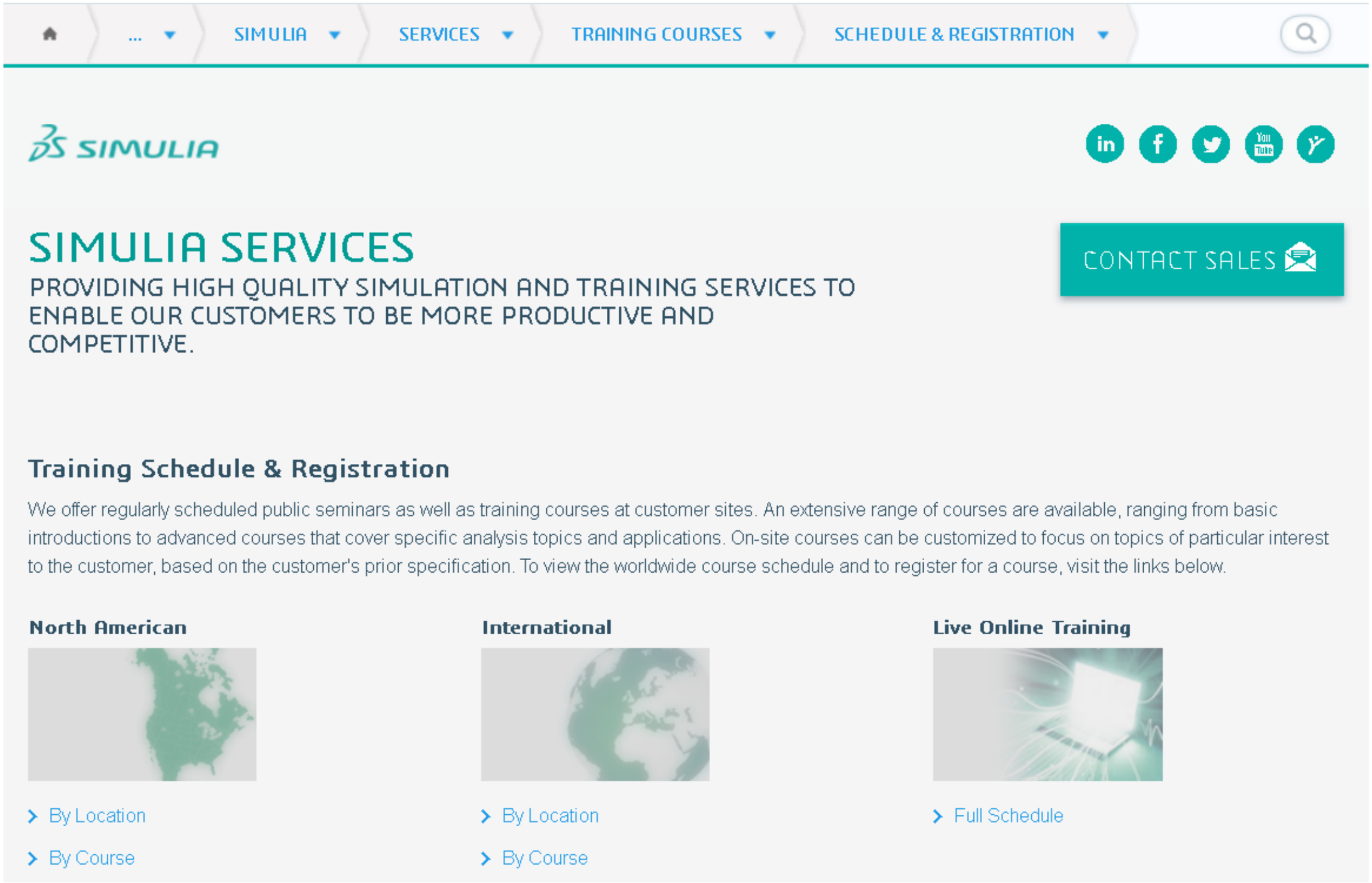
Discover new ways to explore how to leverage realistic simulation to drive product innovation. Join the thousands of Abaqus and Isight users who are already gaining valuable knowledge from the SIMULIA Learning Community.

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 |
| 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 |

# Lesson 1: Introduction to Computational Electromagnetics

***Lesson content:***

- Motivation
- Basics of Electromagnetism
- Computational Electromagnetics in Abaqus
- Workflow of an Electromagnetic Analysis
- Examples



**45 minutes**

# Lesson 2: Geometry, Material Properties, Elements and Meshing

***Lesson content:***

- Geometry Creation
- Material Properties
- Element Technology
- Meshing
- Workshop Preliminaries
- Workshop 1: Heating of a Rod: Problem setup
- Workshop 2: Sphere in a Magnetic Field: Problem setup

**2 hours**

# Lesson 3: Loads and Boundary Conditions

***Lesson content:***

- Introduction
- Loads
- Boundary Conditions
- Symmetry
- Motion

**1 hour**

# Lesson 4: Output and Transfer of Results

***Lesson content:***

- Analysis Procedures
- Co-simulation
- Sequential Mapping
- Output
- Workshop 1 (cont'd): Heating of a Rod: Thermal Response
- Workshop 2 (cont'd): Sphere in a Magnetic Field: Electromagnetic Response
- Workshop 3: Magnetostatic Analysis of a Solenoid Valve
- Workshop 4: Magnetic Pulse Forming of a Metallic Tube

**3 hours**