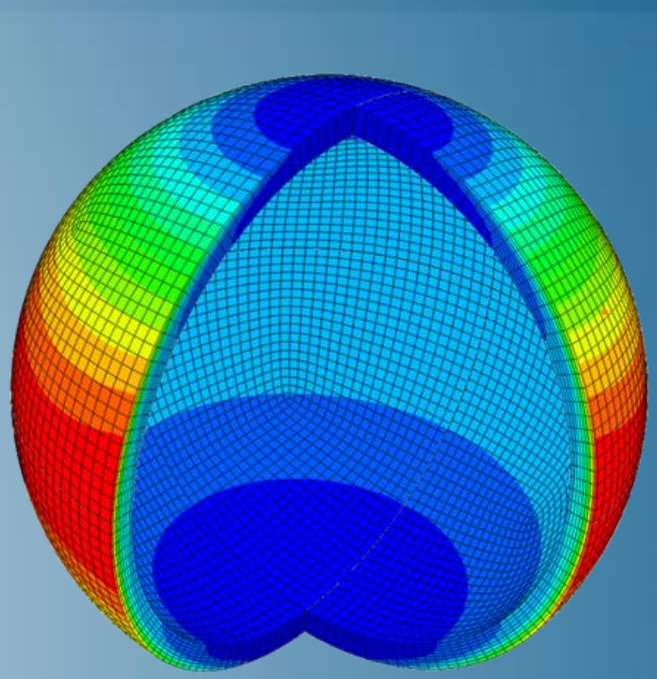
Electromagnetic Analysis with Abaqus

Abaqus 2019



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

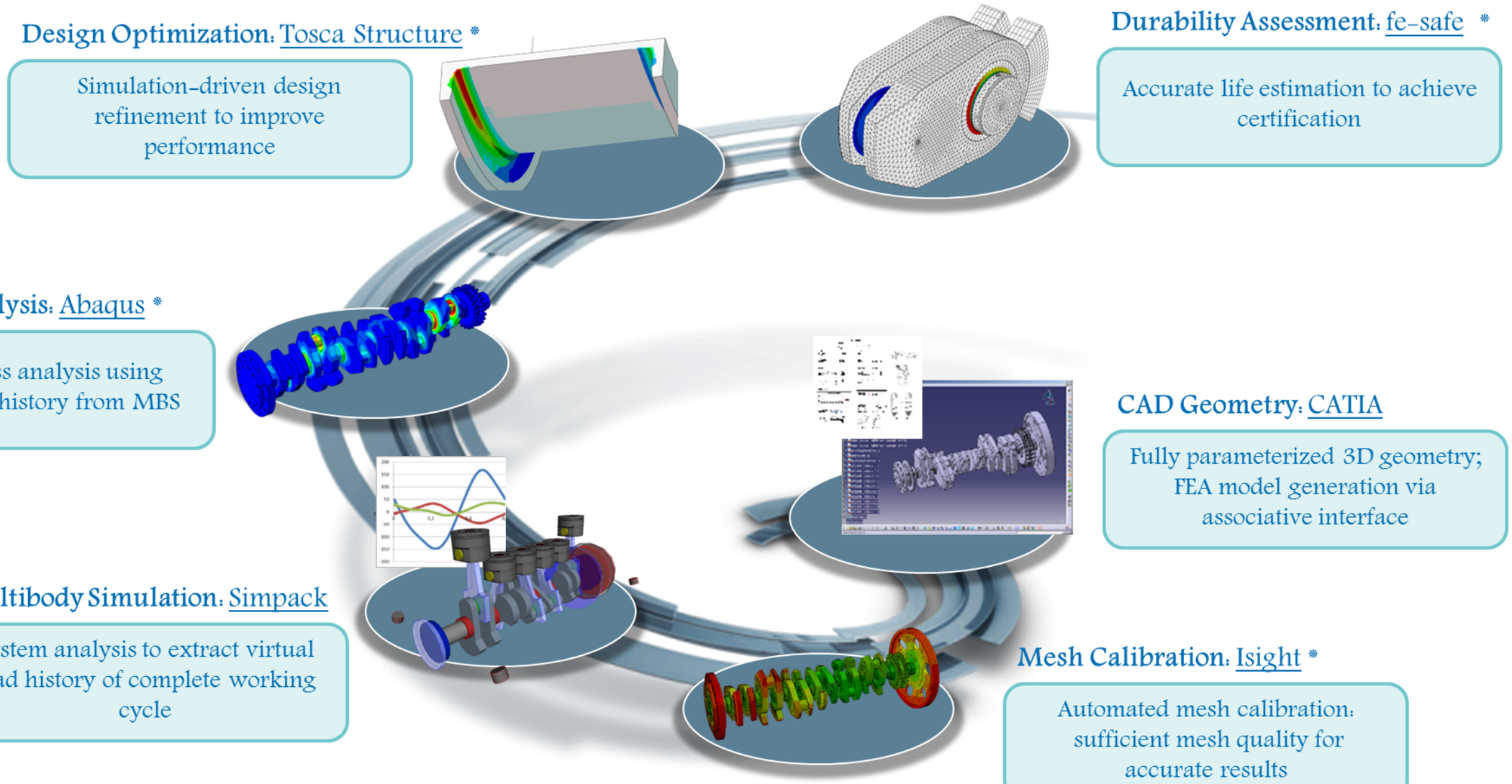
1 day

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 *

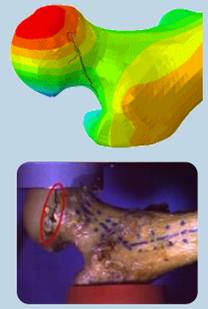
Automated mesh calibration: sufficient mesh quality for accurate results

* Included in extended licensing pool

SIMULIA's Power of the Portfolio

Abaqus

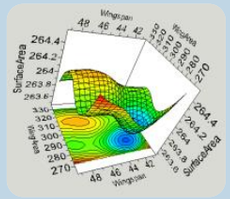
- Routine and Advanced Simulation
- Linear and Nonlinear, Static and Dynamic
- Thermal, Electrical, Acoustics
- Extended Physics through Co-simulation
- Model Preparation and Visualization



Realistic Human Simulation
High Speed Crash & Impact
Noise & Vibration

Isight

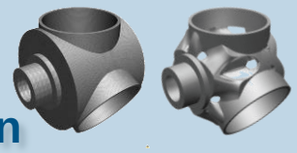
- Process Integration
- Design Optimization
- Parametric Optimization
- Six Sigma and Design of Experiments



Material Calibration
Workflow Automation
Design Exploration

Tosca

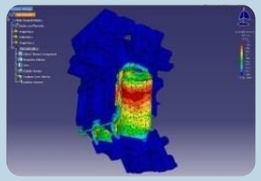
- Non-Parametric Optimization
- Structural and Fluid Flow Optimization
- Topology, Sizing, Shape, Bead Optimization



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

fe-safe

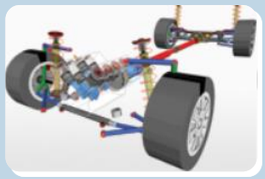
- Durability Simulation
- Low Cycle and High Cycle Fatigue
- Weld, High Temperature, Non-metallics



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

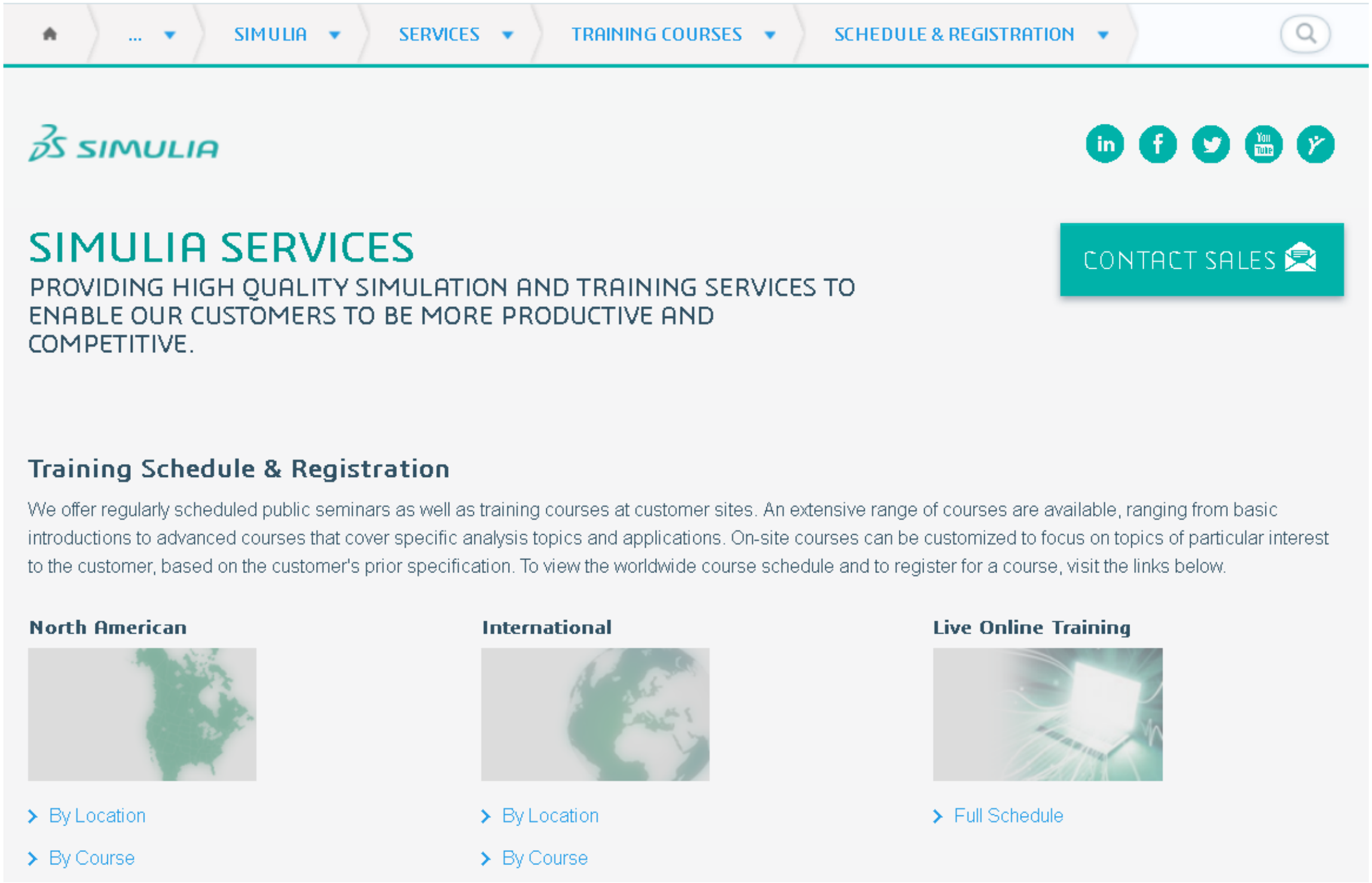
Discover new ways to explore how to leverage realistic simulation to drive product innovation. Join the thousands of Abaqus and Isight users who are already gaining valuable knowledge from the SIMULIA Learning Community.

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	11/18	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
Workshop 1	11/18	Updated for Abaqus 2019
Workshop 2	11/18	Updated for Abaqus 2019
Workshop 3	11/18	Updated for Abaqus 2019
Workshop 4	11/18	Updated for Abaqus 2019

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