DASSAULT SYSTEMES | The 3DEXPERIENCE® Company

Course Catalog

SIMULIA Abaqus

01 September 2016

3DEXPERIENCE®

3DS Learning Solutions | Course Catalog

SIMULIA

SIMULIA

Abaqus

Abaqus/CAE: Geometry Import and Meshing (CAGIM)

Course Code	SIM-en-CAGIM-A-V30R2016
Available Release	2016
Duration	16 hours
Course Material	English
Level	Advanced
Audience	Simulation Analysts
Description	This course provides an in-depth look at several advanced Abaqus/CAE capabilities: CAD geometry import and repair, meshing and partitioning of complicated geometry.
Objectives	Upon completion of this course you will be able to: - Import, edit, and repair CAD geometry. - Import and edit orphan meshes. - Use virtual topology to ease the meshing of complicated geometry. - Partition geometry to enable different meshing techniques.
Prerequisites	None
Available Online	Yes

Abaqus/Explicit: Advanced Topics (ADXP)	
Course Code	SIM-en-ADXP-A-V30R2016
Available Release	2016
Duration	24 hours
Course Material	English
Level	Advanced
Audience	Simulation Analysts
Description	The course emphasizes practical skills and techniques that are needed for analyses with Abaqus/Explicit.
Objectives	Upon completion of this course you will be able to: - Use the explicit dynamics method effectively, including the application of general contact, mass scaling, and adaptive remeshing - Use Abaqus/Explicit and Abaqus/Standard together to solve difficult problems, including results transfer and co-simulation - Model high-strain-rate deformation and failure - Filter output
Prerequisites	This course is recommended for engineers with experience using Abaqus
Available Online	Yes

Abaqus for Offshore Analysis (OFFSH)

Course Code	SIM-en-OFFSH-A-V30R2016
Available Release	2016
Duration	16 hours
Course Material	English
Level	Advanced
Audience	This course is recommended for engineers with experience using Abaqus who work in the Oil and Gas industry.
Description	This course was designed by SIMULIA UK in support of their offshore customers to provide them a more in-depth, industry-specific training. The workshops in this course are completely new and were developed from customer applications.
Objectives	The topics covered in this course include: - Review nonlinear material behavior (metal plasticity and hyperelasticity) - Capabilities of Abaqus element types in general - Specific element discussions include drag chain, pipe, PSI and ITT elements - Pipe-soil interaction, including lateral buckling of a pipe line on a seabed - Abaqus/Aqua capabilities in Abaqus/Standard to model wave, buoyancy, current & wind loading - Coupled Eulerian-Lagrangian (CEL) approach in Abaqus/Explicit
Prerequisites	None
Available Online	Yes

Adaptive Remeshing in Abaqus/Standard (ADAP)	
Course Code	SIM-en-ADAP-A-V30R2016
Available Release	2016
Duration	8 hours
Course Material	English
Level	Advanced
Audience	Simulation Analysts
Description	This course provides an in-depth coverage of the Abaqus features which address adaptive remeshing for solution accuracy.
Objectives	Upon completion of this course you will be able to: - Set up an adaptive remeshing process - Specify adaptive remeshing rules - Interpret error indicators and their associated output variables
Prerequisites	This course is recommended for engineers with experience using Abaqus
Available Online	Yes

Advanced Abaqus Scripting (SCRPT)

Course Code	SIM-en-SCRPT-A-V30R2016
Available Release	2016
Duration	16 hours
Course Material	English
Level	Advanced
Audience	Simulation Analysts
Description	This seminar is offered as a follow-up to the Introduction to Abaqus Scripting course. It is a deeper dive into both Python and the Abaqus Scripting Interface and gives users more hands on exposure with practically oriented workshops of moderate complexity. This course also provides pointers for more specialized and advanced topics.
Objectives	Upon completion of this course you will be able to: - Help students to develop a high level understanding of the Abaqus scripting capabilities and gain some proficiency. - Organize and present the deeper technical details of Python and the Abaqus Scripting Interface. - Expose the strengths and weaknesses of Abaqus scripting. - Encourage the student to use scripting in new ways. - This advanced seminar will take a deeper dive into: - The Abaqus Scripting Interface (ASI) - The core functionality of the Python language and libraries
Prerequisites	Experience scripting with Python and Abaqus is recommended.

Advanced Abaqus Scripting (SCRPT)

Available Online	Yes

Analysis of Composite Materials with Abaqus (MAT)	
Course Code	SIM-en-MAT-A-V30R2016
Available Release	2016
Duration	24 hours
Course Material	English
Level	Advanced
Audience	Simulation Analysts
Description	Composite materials are used in many design applications because of their high stiffness-to-weight ratios. This seminar shows you how to use Abaqus effectively to model composite materials.
Objectives	Upon completion of this course you will be able to: - Define anisotropic elasticity for combining the fiber-matrix response - Define composite layups - Model progressive damage and failure in composites - Model delamination and low-cycle fatigue of composite structures - Model sandwich composite structures and stiffened composite panels
Prerequisites	This course is recommended for engineers with experience using Abaqus
Available Online	Yes

Analysis of Geotechnical Problems with Abaqus (GEOT)

Course Code	SIM-en-GEOT-A-V11R15
Available Release	2016
Duration	16 hours
Course Material	English
Level	Advanced
Audience	This seminar is recommended for engineers with experience using Abaqus/Standard.
Description	Participants are given an overview of modeling geotechnical problems. Experimental testing and how it relates to the calibration of constitutive models for geotechnical materials is reviewed. The seminar teaches users how to use and calibrate the different geotechnical material constitutive models available in Abaqus and discusses the limitations of these models. The coupling between fluid flow and stress/deformation in the analysis of porous media is also considered. Modeling issues related to geotechnical problems are addressed and numerous illustrative examples are examined
Objectives	Upon completion of this course you will be able to: - An overview of modeling geotechnical problems - Experimental testing and how it relates to the calibration of constitutive models for geotechnical materials - How to use and calibrate the different geotechnical material constitutive models available in Abaqus - The limitations of these models - The coupling between fluid flow and stress/ deformation in the analysis of porous media - Modeling issues related to geotechnical problems

Analysis of Geotechnical Problems with Abaqus (GEOT)

Prerequisites	None
Available Online	Yes

Automotive NVH with Abaqus

Course Code	cb0ad5ab-da6c-4fd3-bc17-15c1e7c130c4
Available Release	2016
Duration	24 hours
Course Material	English
Level	Fundamental
Audience	Simulation Analysts
Description	This course focuses on applying the linear dynamics capabilities in Abaqus to NVH-related simulation.
Objectives	Upon completion of this course you will be able to: - Perform natural frequency extractions - Perform sound radiation analyses (acoustics) - Include nonlinear preloading effects in your NVH simulations - Perform Brake squeal analyses - Create constraints and connections for Automotive NVH models - Use substructuring techniques to run your NVH simulations more efficiently - Perform advanced NVH postprocessing (via plug-ins)
Prerequisites	This course is recommended for engineers with experience using Abaqus
Available Online	Yes

Buckling, Postbuckling and Collapse Analysis (BUCK)	
Course Code	SIM-en-BUCK-A-V30R2016
Available Release	2016
Duration	16 hours
Course Material	English
Level	Advanced
Audience	Simulation Analysts
Description	This course blends the theoretical background on such topics as geometric nonlinearity and the Riks method together with examples, guidelines and workshops.
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
Prerequisites	This course is recommended for engineers with experience using Abaqus
Available Online	Yes

Composites Modeler for Abaqus/CAE (CMA)

Course Code	SIM-en-CMA-A-V30R2016
Available Release	2016
Duration	16 hours
Course Material	English
Level	Advanced
Audience	This is an advanced seminar for users who are already familiar with the native Abaqus/CAE composites modeling functionality.
Description	This is an advanced seminar for users who are already familiar with the native Abaqus/CAE composites modeling functionality. Therefore, the Analysis of Composite Materials with Abaqus seminar is recommended as a prerequisite. At the very least, attendees should be familiar with the Abaqus/CAE composite layup functionality. Attendees should also be comfortable postprocessing the results of composites simulations using Abaqus/CAE. An understanding of how composites are manufactured is also helpful.
Objectives	In this course you will learn about: - Composites Modeler for Abaqus/CAE, an add-on product to Abaqus/CAE - How to use Composites Modeler for Abaqus/CAE to account for accurate fiber angles and ply thicknesses in Abaqus simulations to achieve unprecedented accuracy - How to review and quickly modify your composites models to iteratively improve your designs - How to use your composites model to generate manufacturing data thereby ensuring that the analyzed model closely corresponds to the real structure

Composites Modeler for Abaqus/CAE (CMA)	
Prerequisites	The Analysis of Composite Materials with Abaqus seminar is recommended as a prerequisite. At the very least, attendees should be familiar with the Abaqus/CAE composite layup functionality. Attendees should also be comfortable post-processing the results of composites simulations using Abaqus/CAE. An understanding of how composites are manufactured is also helpful.
Available Online	Yes

Co-simulation with Abaqus and Dymola (DYM)	
Course Code	SIM-en-DYM-A-V30R2016
Available Release	2016
Duration	4 hours
Course Material	English
Level	Advanced
Audience	Simulation Analysts
Description	Abaqus-Dymola co-simulation is useful when logical modeling needs to be included in a physical system simulation; for example, it can be used to couple Anti-lock Braking System (ABS) logic modeled in Dymola with an Abaqus rolling tire and brake simulation.
Objectives	Upon completion of this course you will be able to: - Set up an Abaqus model for Abaqus-Dymola co-simulation - Create a simple control system in Dymola - Run a co-simulation
Prerequisites	None
Available Online	Yes

Crashworthiness Analysis with Abaqus (CRASH)	
Course Code	SIM-en-CRASH-A-V30R2016
Available Release	2016
Duration	24 hours
Course Material	English
Level	Advanced
Audience	New and experienced users of Abaqus who will perform structural crashworthiness or occupant safety simulations.
Description	This course is the ideal way to obtain a working knowledge of how to use Abaqus for crashworthiness analysis.
Objectives	Upon completion of this course you will be able to: - Abaqus fundamentals and input syntax - General "automatic" contact modeling - Element selection for crash simulation - Constraints and connections modeling - Material models used in crash simulation - Multiple mechanism damage and failure modeling
Prerequisites	No previous knowledge of Abaqus is required, but knowledge of finite elements and engineering mechanics is necessary.
Available Online	Yes

CZone for Abaqus (CZA)	
Course Code	SIM-en-CZA-A-V30R2016
Available Release	2016
Duration	4 hours
Course Material	English
Level	Advanced
Audience	Engineers with experience using Abaqus/Explicit
Description	By attending this half-day class you will learn how to include crushable composite structures in your impact simulations.
Objectives	Upon completion of this course you will be able to: - Include crushable composite structures in your impact simulations - Understand guidelines for defining crushable composite materials based on composite coupon and component testing - Incorporate crushable composite structures into your models and how to postprocess CZA analysis results
Prerequisites	The Abaqus/Explicit: Advanced Topics and Analysis of Composite Materials with Abaqus seminars are recommended as prerequisites
Available Online	Yes

Electromagnetic Analysis with Abaqus (EMAG)

Course Code	SIM-en-EMAG-A-V30R2016
Available Release	2016
Duration	8 hours
Course Material	English
Level	Advanced
Audience	Simulation Analysts
Description	Abaqus provides computational electromagnetic capabilities for the simulation of problems involving steady-state electrical conduction, piezoelectric phenomena and low-frequency eddy currents. In this course, you will learn how to analyze low frequency eddy current problems in Abaqus/Standard.
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
Prerequisites	This course is recommended for engineers with experience using Abaqus
Available Online	Yes

Element Selection in Abaqus (ELEMC)

Course Code	SIM-en-ELEMC-A-V30R2016
Available Release	2016
Duration	16 hours
Course Material	English
Level	Advanced
Audience	Simulation Analysts
Description	This course provides a brief overview of the distinguishing characteristics of the wide range of continuum and structural elements available in Abaqus for stress analyses. It explains modeling features that may cause certain types of elements to behave poorly.
Objectives	Upon completion of this course you will be able to: - Understand the distinguishing characteristics of the wide range of continuum and structural elements available in Abaqus for stress analyses - Understand modeling features that may cause certain types of elements to behave poorly - Choose appropriate element types for different applications including the effects of fully or nearly incompressible material behavior, contact, bending, etc.
Prerequisites	This course is recommended for engineers with experience using Abaqus
Available Online	Yes

Flexible Multibody Systems with Abaqus (FLEX)	
Course Code	SIM-en-FLEX-A-V30R2016
Available Release	2016
Duration	16 hours
Course Material	English
Level	Advanced
Audience	Simulation Analysts
Description	The goals of this course are to explore the variety of connection types available in Abaqus and to understand how to define connections that suit your needs.
Objectives	The topics include: - Comparison of connectors and MPCs - Basic connector components - Assembled kinematic connections - Local relative displacements and rotations - Defining stops and locks - Defining connector friction - Connector failure - Actuating components of relative motion - Sensors and actuators - Output and postprocessing
Prerequisites	This course is recommended for engineers with experience using Abaqus
Available Online	Yes

FSI Simulation Using Abaqus and 3rd Party Codes (FSI)

Course Code	SIM-en-FSI-A-V30R2016
Available Release	2016
Duration	16 hours
Course Material	English
Level	Advanced
Audience	This seminar is recommended for both structural and CFD engineers with an interest in evaluating and analyzing real world FSI applications.
Description	This seminar provides an introduction to the FSI capability using 3rd-party CFD codes, with an emphasis on enabling users to get started utilizing the capability effectively. Several examples using both Abaqus/Standard and Abaqus/Explicit are used throughout the seminar to illustrate the types of problems that can be solved.
Objectives	Upon completion of this course you will be able to: - Evaluate FSI applications - Create compatible FE and CFD models for FSI - Run FSI problems - Develop co-simulation strategies - Use time incrementation options
Prerequisites	None
Available Online	Yes

GUI Customization with Abaqus (GUIC)

Course Code	SIM-en-GUIC-A-V30R2016
Available Release	2016
Duration	24 hours
Course Material	English
Level	Advanced
Audience	Users interested in modifying and extending the capabilities of Abaqus by customizing their Abaqus interface
Description	This course introduces the Abaqus GUI Toolkit through a combination of lectures, examples and workshops.
Objectives	The goal of this course is to train you to use the Abaqus GUI Toolkit to customize the Abaqus/CAE interface or build your own applications - Learn how to build dialogs and issue commands from the GUI - Learn how to create and modify GUI modules and toolsets - Learn how to create custom applications
Prerequisites	This seminar assumes prior knowledge of the Python programming language and the Abaqus kernel commands. Thus, students must attend the Introduction to Abaqus Scripting seminar prior to attending this class. Experience with object-oriented programming and GUI toolkits is recommended, but not required.
Available Online	Yes

Heat Transfer and Thermal-Stress Analysis with Abaqus (HEAT)

Course Code	SIM-en-HEAT-A-V30R2016
Available Release	2016
Duration	16 hours
Course Material	English
Level	Advanced
Audience	Simulation Analysts
Description	The success of many structural designs requires a thorough understanding of both the thermal and mechanical response of the design. Temperature-dependent material properties, thermally-induced deformation, and temperature variations all may be important design considerations.
Objectives	Upon completion of this course you will be able to: - Perform steady-state and transient heat transfer simulations - Solve cavity radiation problems - Model latent heat effects - Perform adiabatic, sequential, and fully coupled thermal-stress analyses - Model contact in heat transfer problems
Prerequisites	This course is recommended for engineers with experience using Abaqus
Available Online	Yes

Introduction to Abaqus (ABI)	
Course Code	SIM-en-ABI-F-V30R2016
Available Release	2016
Duration	40 hours
Course Material	English
Level	Fundamental
Audience	Simulation Analysts
Description	This course is a comprehensive and unified introduction to the modeling and analysis capabilities of Abaqus. It teaches you how to solve linear and nonlinear problems, submit and monitor analysis jobs and view simulation results using the interactive interface of Abaqus.
Objectives	Upon completion of this course you will be able to: - Use Abaqus/CAE to create complete finite element models. - Use Abaqus/CAE to submit and monitor analysis jobs. - Use Abaqus/CAE to view and evaluate simulation results. - Solve structural analysis problems using Abaqus/Standard and Abaqus/Explicit, including the effects of material nonlinearity, large deformation and contact.
Prerequisites	None
Available Online	Yes

Introduction to Abaqus/CAE (ICAE)

Course Code	SIM-en-ICAE-F-V30R2016
Available Release	2016
Duration	16 hours
Course Material	English
Level	Fundamental
Audience	Simulation Analysts
Description	Abaqus/CAE provides a complete interactive environment for creating Abaqus models, submitting and monitoring analysis jobs and viewing and manipulating simulation results.
Objectives	Upon completion of this course you will be able to: - Use Abaqus/CAE to create complete finite element models. - Use Abaqus/CAE to submit and monitor analysis jobs. - Use Abaqus/CAE to view and evaluate simulation results
Prerequisites	None
Available Online	Yes

Introduction to Abaqus/CFD for Multiphysics Applications (CFD)

Course Code	SIM-en-CFD-F-V30R2016
Available Release	2016
Duration	16 hours
Course Material	English
Level	Fundamental
Audience	Simulation Analysts
Description	Abaqus/CFD provides advanced computational fluid dynamics capabilities with extensive support for preprocessing and postprocessing provided in Abaqus/CAE. In this course, you will learn how to analyze CFD problems in Abaqus, as well as nonlinear coupled fluid-thermal and fluid-structural problems.
Objectives	Upon completion of this course you will be able to: - Set up and create CFD and FSI models with Abaqus - Perform CFD analyses with Abaqus - Perform FSI analyses with Abaqus - Postprocess CFD and FSI results
Prerequisites	None
Available Online	Yes

Introduction to Abaqus/Standard and Abaqus/Explicit (IABA)	
Course Code	SIM-en-IABA-F-V30R2016
Available Release	2016
Duration	24 hours
Course Material	English
Level	Fundamental
Audience	Simulation Analysts
Description	This introductory course is the ideal way to obtain a working knowledge of how to use both Abaqus/ Standard and Abaqus/Explicit to solve linear and nonlinear problems. The seminar introduces you to the analysis capabilities of Abaqus using the keywords interface.
Objectives	Upon completion of this course you will be able to: - Complete finite element models using Abaqus keywords. - Submit and monitor analysis jobs. - View and evaluate simulation results. - Solve structural analysis problems using Abaqus/ Standard and Abaqus/Explicit, including the effects of material nonlinearity, large deformation and contact.
Prerequisites	None
Available Online	Yes

Introduction to Abaqus Scripting (ISRPT)	
Course Code	SIM-en-ISRPT-A-V30R2016
Available Release	2016
Duration	16 hours
Course Material	English
Level	Advanced
Audience	Simulation Analysts
Description	This seminar covers basic usage of the Abaqus Scripting Interface and Python's syntax. It includes numerous hands-on exercises for the student to learn to automate tasks that are common to most analysts.
Objectives	Upon completion of this course you will be able to: - Help students to develop a high level understanding of the Abaqus scripting capabilities. - Organize and present the technical details of Python and the Abaqus Scripting Interface. - Expose the strengths and weaknesses of Abaqus scripting. - Encourage the student to use scripting in new ways.
Prerequisites	None
Available Online	Yes

Linear Dynamics with Abaqus (LNDYN)

Course Code	SIM-en-LNDYN-A-V30R2016
Available Release	2016
Duration	16 hours
Course Material	English
Level	Advanced
Audience	Simulation Analysts
Description	This course introduces the user to the algorithms and methods used to study linear dynamic problems with Abaqus/Standard.
Objectives	Upon completion of this course you will be able to: - Extract eigenmodes about a certain frequency - Determine whether the number of extracted eigenmodes is sufficient to represent the structure's response adequately - Perform transient, steady-state, response spectrum and random response analyses using the eigenmodes - Use multiple base motions - Apply damping in linear problems
Prerequisites	This course is recommended for engineers with experience using Abaqus
Available Online	Yes

Metal Forming with Abaqus (METF)

Course Code	SIM-en-METF-A-V30R2016
Available Release	2016
Duration	24 hours
Course Material	English
Level	Advanced
Audience	This course is recommended for engineers with experience using Abaqus
Description	Metal forming processes are highly nonlinear because they involve geometric, material and contact nonlinearities.
Objectives	In this course you will learn practical modeling skills and techniques for: - Stamping - Hydroforming - Punch stretching - Forging - Rolling - Drawing - Superplastic forming
Prerequisites	None
Available Online	Yes

Metal Inelasticity in Abaqus (METAL)	
Course Code	SIM-en-METAL-A-V30R2016
Available Release	2016
Duration	16 hours
Course Material	English
Level	Advanced
Audience	Simulation Analysts
Description	This seminar provides a brief overview of the inelastic behavior observed in metals and the basic concepts of plasticity theory.
Objectives	Upon completion of this course you will be able to: - Metals that show inelastic work hardening - The Bauschinger effect - "Ratchetting" and relaxation of the mean stress under cyclic loading - Strain-rate-dependent inelastic behavior - Temperature-dependent plasticity - Heat generated by plastic deformation - Ductile failure of metallic materials - Plastic behavior in porous and brittle (cast iron) metals - Creep behavior in metals
Prerequisites	This course is recommended for engineers with experience using Abaqus
Available Online	Yes

Modeling Contact with Abaqus/Standard (CONT)

Course Code	SIM-en-CONT-A-V30R2016
Available Release	2016
Duration	16 hours
Course Material	English
Level	Advanced
Audience	Simulation Analysts
Description	Participants are given a brief overview of the contact formulation and contact logic used in Abaqus/ Standard. The hands-on workshops provide ample opportunity to use the concepts developed in the lectures and to learn how to postprocess the results of a contact analysis.
Objectives	Upon completion of this course you will be able to: - Define general contact and contact pairs - Define appropriate surfaces (rigid or deformable) - Model frictional contact - Model large sliding between deformable bodies - Resolve overclosures in interference fit problems
Prerequisites	This course is recommended for engineers with experience using Abaqus/Standard
Available Online	Yes

Modeling Extreme Deformation and Fluid Flow with Abaqus (FLOW)

Course Code	SIM-en-FLOW-A-V30R2016
Available Release	2016
Duration	16 hours
Course Material	English
Level	Advanced
Audience	Simulation Analysts
Description	This seminar aims at providing users with a solid understanding of the Coupled Eulerian-Lagrangian (CEL) and Smoothed Particle Hydrodynamic (SPH) methods and illustrating approaches to setting-up and analyzing real world problems using these advanced analysis methods.
Objectives	Upon completion of this course you will be able to: - Create Eulerian meshes and define the initial material location within an Eulerian mesh - Specify initial conditions, boundary conditions and loads to materials in the Eulerian domain - Use general contact to model Eulerian-Lagrangian interactions - Create SPH meshes - Automatically convert conventional continuum elements to SPH particles - Define initial conditions, boundary conditions, and loads on SPH particles - Define contact interactions between SPH particles an element-based or analytical surfaces - Understand the differences between the CEL, SPH, and CFD approaches

Modeling Extreme Deformation and Fluid Flow with Abaqus (FLOW)	
Prerequisites	This course is recommended for engineers with experience using Abaqus
Available Online	Yes

Modeling Fracture and Failure with Abaqus (FRAC)

Course Code	SIM-en-FRAC-A-V30R2016
Available Release	2016
Duration	24 hours
Course Material	English
Level	Advanced
Audience	Simulation Analysts
Description	Fracture and failure modeling allows for product designs that maximize the safe operating life of structural components. Abaqus offers many capabilities that enable fracture and failure modeling. This seminar provides a detailed discussion of these capabilities.
Objectives	Upon completion of this course you will be able to: - Use proper modeling techniques for capturing crack-tip singularities in fracture mechanics problems - Use Abaqus/CAE to create meshes appropriate for fracture studies - Calculate stress intensity factors and contour integrals around a crack tip - Simulate material damage and failure - Simulate crack growth using cohesive behavior, VCCT, and XFEM - Simulate low-cycle fatigue crack growth
Prerequisites	This course is recommended for engineers with experience using Abaqus
Available Online	Yes

Modeling Rubber and Viscoelasticity with Abaqus (MRUB)

Course Code	SIM-en-MRUB-A-V30R2016
Available Release	2016
Duration	16 hours
Course Material	English
Level	Advanced
Audience	Simulation Analysts
Description	This course provides a brief overview of finite deformations and the material models used for rubber and resilient foam.
Objectives	Upon completion of this course you will be able to: - Use experimental test data to calculate material constants - Check the stability of the Abaqus material model at extreme strains - Obtain the best possible material constants from the available test data - Select elements for modeling rubber and foams - Design an appropriate finite element mesh - Model viscoelastic behavior in both the time and frequency domain - Use a user subroutine to define the hyperelastic behavior
Prerequisites	This course is recommended for engineers with experience using Abaqus
Available Online	Yes

Modeling Stents Using Abaqus (STENT)	
Course Code	SIM-en-STENT-A-V30R2016
Available Release	2016
Duration	16 hours
Course Material	English
Level	Advanced
Audience	Simulation Analysts
Description	This course focuses on the use of Abaqus for modeling and analyzing stents. However, its content can also be useful when modeling other types of medical devices. The course is targeted at engineers responsible for the design of medical devices who are looking to accelerate their understanding of the highly complex mechanical behavior associated with performance of such devices.
Objectives	Upon completion of this course you will be able to: - Create geometry for modeling stents and tools - Choose the proper element type - Choose material models: elastic-plastic (Stainless Steel), superelastic-plastic (Nitinol), hyperelastic (vessels) - Perform stent analyses: Static, Implicit and Explicit Dynamics - Define contact and constraints - Postprocess stent analyses - Perform fatigue evaluation
Prerequisites	This course is recommended for engineers with experience using Abaqus
Available Online	Yes

Non-parametric Optimization with Abaqus (OPT)

Course Code	SIM-en-OPT-A-V30R2016
Available Release	2016
Duration	8 hours
Course Material	English
Level	Advanced
Audience	Finite element analysts or product designers with some background in finite element analysis
Description	The objective of this seminar is to introduce users to the non-parametric optimization capabilities available in “Tosca for Abaqus.” An easy to use interface native to Abaqus/CAE is available for the setup, execution, monitoring and postprocessing of topology, shape and sizing optimization problems. In combination with Abaqus analysis products, Tosca for Abaqus offers an unparalleled structural optimization capability for highly nonlinear problems.
Objectives	Upon completion of this course you will be able to: - Apply topology, shape, sizing and bead optimization techniques to your designs and produce lightweight, strong and durable components - Reduce iterations of designs - Use the optimization interface in Abaqus/CAE for setup, execution, monitoring and postprocessing of topology, shape, sizing and bead optimization problems - Use Tosca’s structural optimization capability for highly nonlinear problems
Prerequisites	Some familiarity with Abaqus/CAE is useful but not required.

Non-parametric Optimization with Abaqus (OPT)

Available Online	Yes

Obtaining a Convergenced Solution with Abaqus (CONV)

Course Code	SIM-en-CONV-A-V30R2016
Available Release	2016
Duration	16 hours
Course Material	English
Level	Advanced
Audience	Simulation Analysts
Description	Obtaining converged solutions for highly nonlinear simulations can sometimes be challenging. Difficulties can arise, especially in simulations involving contact, complicated material models and geometrically unstable behavior. Many years of practical experience in understanding and resolving convergence issues have been condensed into this course.
Objectives	Upon completion of this course you will be able to: - Understand how nonlinear problems are solved in Abaqus - Develop Abaqus models that will converge - Identify modeling errors that cause models to experience convergence difficulties - Recognize when a problem is too difficult or too ill-posed to be solved effectively
Prerequisites	This course is recommended for engineers with experience using Abaqus
Available Online	Yes

Structural-Acoustic Analysis Using Abaqus (ACOU)	
Course Code	SIM-en-ACOU-A-V30R2016
Available Release	2016
Duration	16 hours
Course Material	English
Level	Advanced
Audience	Simulation Analysts
Description	This seminar covers the fundamentals of acoustics phenomena and then shows how to use Abaqus to solve a wide range of acoustics problems.
Objectives	Upon completion of this course you will be able to: - Pure acoustics analysis - Coupled structural-acoustic analysis - Scattering and shock analysis - Mesh size and mesh density effects for different analysis procedures - Acoustic analysis output and postprocessing
Prerequisites	This course is recommended for engineers with experience using Abaqus. Some understanding of acoustics is helpful but is not required.
Available Online	Yes

Substructures and Submodeling with Abaqus (SUPSUB)

Course Code	SIM-en-SUPSUB-A-V30R2016
Available Release	2016
Duration	16 hours
Course Material	English
Level	Advanced
Audience	Simulation Analysts
Description	The size and complexity of designs that are analyzed and tested with Abaqus continues to grow. Substructures and submodeling are two effective techniques that allow the analyst to study problems that are too large to simulate with a conventional modeling approach.
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
Prerequisites	This course is recommended for engineers with experience using Abaqus
Available Online	Yes

Tire Analysis with Abaqus: Advanced Topics (TIRE2)	
Course Code	SIM-en-TIRE2-A-V30R2016
Available Release	2016
Duration	8 hours
Course Material	English
Level	Advanced
Audience	This course is recommended for tire analysts with experience using Abaqus
Description	Modern tires are among the most complex structures in production and their complexities span a broad range of the capabilities available in Abaqus. This seminar covers topics addressing advanced tire modeling techniques and serves as a follow-up to the Tire Analysis with Abaqus: Fundamentals course.
Objectives	Topics covered in this course include: - Steady-state rolling using Eulerian techniques in Abaqus/Standard - Hydroplaning simulation using Coupled Eulerian-Lagrangian technique - Efficient steady-state dynamics analysis - Transient analysis using Abaqus/Explicit - Substructuring and submodeling
Prerequisites	Tire Analysis with Abaqus: Fundamentals
Available Online	Yes

Tire Analysis with Abaqus: Fundamentals (TIRE)

Course Code	SIM-en-TIRE-F-V30R2016
Available Release	2016
Duration	16 hours
Course Material	English
Level	Fundamental
Audience	This course is recommended for tire analysts with experience using Abaqus
Description	Modern tires are among the most complex structures in production and their complexities span a broad range of the capabilities available in Abaqus. Since tire modeling is a specialized field, this seminar covers the many important yet basic capabilities in Abaqus that are specifically relevant to tire modeling.
Objectives	In this course you will learn about: - Choosing appropriate elements - Methods of modeling reinforcement - Contact modeling details pertinent to tire modeling - Fundamentals of material modeling-stress and strain measures, material directions - Linear elasticity, hyperelasticity and viscoelasticity - Efficient axisymmetric to three-dimensional model generation and results transfer
Prerequisites	None
Available Online	Yes

Writing User Subroutines with Abaqus (SUBR)

Course Code	SIM-en-SUBR-A-V30R2016
Available Release	2016
Duration	16 hours
Course Material	English
Level	Advanced
Audience	This course is recommended for engineers with experience using Abaqus.
Description	This course explains when to consider the use of such user subroutines and how to approach their development. Detailed descriptions are given of the data required for these subroutines, the additional statements to be included and the variables that are available within the routine. Particular attention is paid to highlighting good practice in user subroutine development.
Objectives	In this course you will learn about: - When and how to use subroutines - DLOAD, VDLOAD, and UTRACLOAD for specifying user-defined loading - FILM for specifying user-defined film conditions - USDFLD and VUSDFLD for defining field variable dependence - UVARM for defining a user output variable - UHYPER for modeling hyperelastic materials - UMAT and VUMAT for allowing constitutive models to be added to the program - UEL and VUEL for allowing the creation of user-defined elements
Prerequisites	A working knowledge of the finite element method and programming in either Fortran or C

Writing User Subroutines with Abaqus (SUBR)	
Available Online	Yes

SIMULIA

fe-safe

Automating Analysis in fe-safe (AAFS)

Course Code	SIM-en-AAFS-A-V30R2016
Available Release	2016
Duration	8 hours
Course Material	English
Level	Advanced
Audience	Simulation Analysts
Description	In this course you will learn how to extend typical GUI analysis in fe-safe to include different methods of automated fatigue analysis.
Objectives	Upon completion of this course you will be able to: - Set up and run various automated fatigue analyses using fe-safe - Package a project directory used with fe-safe to replicate analysis - Initiate analysis from a command prompt - Make use of full read and pre-scan to open FEA models - Troubleshoot and customize batch files and macros - Automate analysis in process integration tools - Submit analysis for distributed processing
Prerequisites	Introduction to fe-safe
Available Online	Yes

Introduction to fe-safe (IFES)	
Course Code	SIM-en-IFES-F-V30R2016
Available Release	2016
Duration	16 hours
Course Material	English
Level	Fundamental
Audience	Simulation Analysts
Description	In this practical introduction to fe-safe you will learn how to set up and run various fatigue analyses using fe-safe. The course includes many hands-on tutorials and practical examples.
Objectives	Upon completion of this course you will be able to: - Set up and run various fatigue analyses using fe-safe - Set up models and import models into fe-safe - Select a material for fatigue analysis - Set up your loadings - Run various analyses in fe-safe
Prerequisites	None
Available Online	Yes

SIMULIA

Isight

Introduction to Isight (ISGT)

Course Code	SIM-en-ISGT-F-V30R2016
Available Release	2016
Duration	16 hours
Course Material	English
Level	Fundamental
Audience	Simulation Analysts
Description	Isight is a Process Integration and Design Optimization (PIDO) software framework, which enables various applications to be easily integrated. With Isight you can create flexible simulation process flows to automate the exploration of design alternatives and identification of optimal performance parameters. This course comprehensively covers the Design and Runtime Gateways along with several fundamental components, exposing users to the ways in which a workflow can be built in Isight and the ways in which the design space can be explored.
Objectives	Upon completion of this course you will be able to: - Automate a series of functions to create a Sim-flow - Add components to a Sim-flow - Set up the core component - Configure components to pass data to/from each other - Execute a Sim-flow - Visualize Sim-flow results - Evaluate design alternatives - Create a Sim-flow to capture a process, by integrating various software in the company. - Perform Design Optimization and gain Design Space understanding by using various techniques such as DOE, Optimization, Monte Carlo etc.

Introduction to Isight (ISGT)

Prerequisites	None
Available Online	Yes

Isight Component Development (ISCD)	
Course Code	SIM-en-ISCD-A-V30R2016
Available Release	2016
Duration	16 hours
Course Material	English
Level	Advanced
Audience	Simulation Analysts
Description	This course covers the process of designing, building, publishing, debugging and testing custom components and plug-ins, utilizing the Isight SDK. The course is highly interactive with a strong emphasis on practical workshops using a standard Integrated Development Environment (IDE).
Objectives	Upon completion of this course you will be able to: - Understand component requirements - Develop component packages for Isight
Prerequisites	This course is recommended for engineers with experience using Isight
Available Online	Yes

SIMULIA

Tosca Fluid

Introduction to Tosca Fluid (TOSCFL)	
Course Code	SIM-en-TOSCFL-A-V30R2016
Available Release	2016
Duration	8 hours
Course Material	English
Level	Advanced
Audience	CFD Analysts working with STAR-CD or ANSYS Fluent
Description	This course is a comprehensive introduction to the fluid optimization capabilities of Tosca Fluid. Attendees will learn how to define and solve basic topology optimization tasks for internal flow problems, submit optimization jobs, and view and evaluate the results.
Objectives	Upon completion of this course you will be able to: - Solve fundamental topology optimization problems for internal flow applications - Postprocess results and perform surface smoothing - Follow-up and transfer results into the CAE-environment
Prerequisites	Basic familiarity with CFD
Available Online	Yes

SIMULIA

Tosca Structure

Introduction to Tosca Structure (TOSCST)	
Course Code	SIM-en-TOSCST-F-V30R2016
Available Release	2016
Duration	16 hours
Course Material	English
Level	Fundamental
Audience	Simulation Analysts independent of FE-Solver and Pre-/Postprocessing environment in use
Description	This course is a comprehensive introduction to the structural optimization capabilities of Tosca Structure.
Objectives	Upon completion of this course you will be able to: - Upon completion of this course you will be able to create optimal design concepts or improve existing designs of mechanical structures: - Solve fundamental topology, shape, sizing and bead optimization problems - Optimize parts regarding weight, stiffness and durability - Visualize, evaluate and transfer optimization results
Prerequisites	None (basic knowledge of finite element analysis)
Available Online	Yes

Tosca Structure nonlinear: Optimization for Structures with Nonlinearity (TOSCNL)	
Course Code	SIM-en-TOSCNL-A-V30R2016
Available Release	2016
Duration	8 hours
Course Material	English
Level	Advanced
Audience	Simulation Analysts, Tosca Structure users
Description	In this course you will learn how to perform structural optimization (topology and shape optimization) based on nonlinear simulation results.
Objectives	Upon completion of this course you will be able to: - Contact - Nonlinear materials - Nonlinear geometries
Prerequisites	Basic knowledge of Tosca Structure
Available Online	Yes

IF WE™