DASSAULT SYSTEMES | The 3DEXPERIENCE® Company

Course Catalog

Learning Experience for SIMULIA Structures SMSTRLX-OC

22 May 2023

3DEXPERIENCE®

Learning Experience | Course Catalog

Learning Experience | Course Catalog

Learning Experience for SIMULIA Structures - SMSTRLX-OC

Abaqus/CAE: Geometry Import and Meshing	
Course Code	SIM-en-CAGIM-A-V30R2023
Available Releases	SIMULIA 2018 , SIMULIA 2019 , SIMULIA 2020 , SIMULIA 2021 , SIMULIA 2022 , SIMULIA 2023 , SIMULIA V6.12
Duration	13.92 hours
Course Material	
Level	Advanced
Audience	This course is recommended for users with a basic knowledge of Abaqus/CAE who wish to become more proficient with the product.
Description	Real-world engineering commonly involves the analysis and design of complicated geometry. These types of analysis depend critically on having a modeling tool with a robust geometry import capability in conjunction with advanced, easy-to-use mesh generation algorithms. This course provides an in-depth look at several advanced Abaqus/CAE capabilities: CAD geometry import and repair, meshing and partitioning of complicated geometry. Both native and neutral geometry formats are discussed. An in-depth treatment of meshing techniques is also provided, including the use of virtual topology to ease the meshing of complicated geometry in the presence of small geometric features. The course consists of lectures, demonstrations and workshops.
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.

Abaqus/CAE: Geometry Import and Meshing	
Prerequisites	Introduction to Abaqus/CAE or equivalent.
Available Online	Yes

Abaqus/Explicit: Advanced Topics	
Course Code	SIM-en-ADXP-A-V30R2023
Available Releases	SIMULIA 2018 , SIMULIA 2019 , SIMULIA 2020 , SIMULIA 2021 , SIMULIA 2022 , SIMULIA 2023 , SIMULIA V6.12
Duration	18.67 hours
Course Material	
Level	Advanced
Audience	This course is recommended for relatively new Abaqus/Explicit users and Abaqus/Standard users who want to learn Abaqus/Explicit, regardless of industry or application.
Description	This course emphasizes practical skills and techniques that are needed for analyses with Abaqus/Explicit. The course uses examples derived from actual industrial applications to reinforce the concepts and issues discussed in the lessons.
Objectives	The topics discussed include the following. - The explicit dynamics method - General contact - Automatic mass scaling for impact problems - Automatic mass scaling for quasi-static problems - Using Abaqus/Explicit and Abaqus/Standard together to solve difficult problems, including results transfer and co-simulation - Modeling high-strain-rate deformation and failure - Output filtering - Managing large models
Prerequisites	Introduction to Abaqus or equivalent.
Available Online	Yes

Abaqus for Offshore Analysis	
Course Code	SIM-en-OFFSH-A-V30R2023
Available Releases	SIMULIA 2018 , SIMULIA 2019 , SIMULIA 2020 , SIMULIA 2021 , SIMULIA 2022 , SIMULIA 2023 , SIMULIA V6.12
Duration	12.67 hours
Course Material	
Level	Advanced
Audience	This course is recommended for engineers with experience using Abaqus who work in the Oil and Gas industry.
Description	The offshore oil and gas industry has some unique analysis challenges. Complex loading conditions, often highly nonlinear stress states and extensive contact requires advanced FEA software and experienced analysts to be successful. This course was designed for Abaqus customers in the Oil and Gas industry to provide them a more in-depth, industry-specific training. This course is composed of lectures, demonstrations and hands-on workshops.
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

Abaqus for Offshore Analysis	
Prerequisites	Introduction to Abaqus or equivalent and experience using Abaqus.
Available Online	Yes

Adaptive Remeshing in Abaqus/Standard	
Course Code	SIM-en-ADAP-A-V30R2023
Available Releases	SIMULIA 2018 , SIMULIA 2019 , SIMULIA 2020 , SIMULIA 2021 , SIMULIA 2022 , SIMULIA 2023 , SIMULIA V6.12
Duration	6.17 hours
Course Material	
Level	Advanced
Audience	This course is recommended for simulation analysts who require adaptive re-meshing to meet specified solution accuracy criteria.
Description	This course provides an in-depth coverage of the Abaqus features which address adaptive remeshing for solution accuracy. Abaqus/CAE and Abaqus/Standard work together to adaptively remesh your model in response to user-specified criteria. These criteria include error indicator and mesh-size targets. Usage can be manual or fully automatic, providing options to control the way the adaptivity process is executed. This capability eliminates the mesh-related uncertainty associated with the analysis by using well-established methods for calculating solution error indicators, which in turn drive the remeshing process, ultimately providing a balance between solution accuracy and cost.
Objectives	The topics discussed include the following. Workshops and example problems are used to illustrate the techniques. - Overview and comparison of adaptivity techniques in Abaqus - Basics of adaptive remeshing - Error indicators and the associated output variables - Specification of adaptive remeshing rules - The adaptive remeshing process

Adaptive Remeshing in Abaqus/Standard	
	- Setting up an adaptive remeshing process
Prerequisites	Introduction to Abaqus or equivalent and experience using Abaqus.
Available Online	Yes

Advanced Abaqus Scripting	
Course Code	SIM-en-SCRPT-A-V30R2023
Available Releases	SIMULIA 2018 , SIMULIA 2019 , SIMULIA 2020 , SIMULIA 2021 , SIMULIA 2022 , SIMULIA 2023 , SIMULIA V6.12
Duration	19.92 hours
Course Material	
Level	Advanced
Audience	This course is recommended for all Abaqus users who have a basic familiarity of scripting and are looking to sharpen their skills.
Description	This course 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	After attending this course, students will be able to: - Write scripts of moderate complexity to automate pre- and postprocessing tasks and improve productivity - Use best practices for maintaining Abaqus scripts and optimizing their performance - Use advanced techniques to take advantage of the Abaqus Object Model - Set up parametric studies using ASI - Build and modify simple GUI plug-ins using the Really Simple GUI (RSG) framework - Leverage built-in features of Python to build applications - Write Python scripts for utility tasks that interface with the operating system and file system

Advanced Abaqus Scripting	
	- Understand how to locate and utilize powerful third-party Python modules - Understand and utilize Python's object-oriented features
Prerequisites	The Introduction to Abaqus Scripting course is highly recommended before attending this seminar. Users proficient with programming (in at least one language) and who are reasonable familiarity with Abaqus/CAE may attend both seminars in series. More experienced Abaqus users already familiar with basics of scripting using Python may attend this advanced seminar directly.
Available Online	Yes

Analysis of Composite Materials with Abaqus	
Course Code	SIM-en-MAT-A-V30R2023
Available Releases	SIMULIA 2018 , SIMULIA 2019 , SIMULIA 2020 , SIMULIA 2021 , SIMULIA 2022 , SIMULIA 2023 , SIMULIA V6.12
Duration	21.25 hours
Course Material	
Level	Advanced
Audience	This course is recommended for simulation analysts whose work involves composite materials.
Description	Composite materials are used in many design applications because of their high stiffness-to-weight ratios. This course 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 using Abaqus/CAE - Achieve the correct material orientation of the layers of composite shells and solid elements - Model sandwich composite structures and stiffened composite panels - Model progressive damage and failure in composites - Model delamination and low-cycle fatigue of composite structures
Prerequisites	Introduction to Abaqus or equivalent and experience using Abaqus.
Available Online	Yes

Analysis of Geotechnical Problems with Abaqus	
Course Code	SIM-en-GEOT-A-V30R2023
Available Releases	SIMULIA 2018 , SIMULIA 2019 , SIMULIA 2020 , SIMULIA 2021 , SIMULIA 2022 , SIMULIA 2023 , SIMULIA V6.12
Duration	11.42 hours
Course Material	
Level	Advanced
Audience	This course is recommended for simulation analysts who will perform geotechnical simulations.
Description	This course provides an overview of modeling geotechnical problems with Abaqus, including issues related to experimental testing, calibration of constitutive models and coupling between fluid flow and stress/deformation in the analysis of porous media. Modeling issues related to geotechnical problems are addressed and numerous illustrative examples are examined.
Objectives	The topics covered in this course include the following: - An overview of modeling geotechnical problems - Experimental testing and how it relates to the calibration of constitutive models for geotechnical materials - Using and calibrating the different geotechnical material constitutive models available in Abaqus and their limitations - The coupling between fluid flow and stress/ deformation in the analysis of porous media - Modeling issues related to geotechnical problems
Prerequisites	Introduction to Abaqus and experience using Abaqus/ Standard.

Analysis of Geotechnical Problems with Abaqus	
Available Online	Yes

Automating Analysis in fe-safe	
Course Code	SIM-en-AAFS-A-V30R2023
Available Releases	SIMULIA 2018 , SIMULIA 2019 , SIMULIA 2020 , SIMULIA 2021 , SIMULIA 2022 , SIMULIA 2023
Duration	5.83 hours
Course Material	
Level	Advanced
Audience	Simulation Analysts
Description	In this course you will learn how to extend fatigue analysis originally configured in the fe-safe GUI to include different methods of automated fatigue analysis. This includes exporting and importing a project archive and using project settings files generated in the fe-safe GUI. Automation can be completed using a command line execution, macro file, or batch file. The different ways of opening the FEA solution will be discussed in the context of these methods. You will learn how to change one or more settings in an fe-safe analysis using each of these methods. The examples include analyses with verity in fe-safe as well as fe-safe/Rubber. Integration with SIMULIA Isight and Tosca Structure will be covered.
Objectives	Upon completion of this course you will be able to: - Set up and run various automated fatigue analyses using fe-safe - Export a project configured in the fe-safe GUI to replicate analysis - Use a full read to import a new FEA solution or refresh the existing solution - Change settings and run analysis from the command prompt - Use pre-scanning and group commands in a macro file - Change settings and run analysis in a macro

Automating Analysis in fe-safe	
	- Execute a macro from the command line or in the fe-safe GUI - Change settings and run analysis in a batch file - Troubleshoot and customize automation of fatigue analyses
Prerequisites	Introduction to fe-safe
Available Online	Yes

Automotive NVH with Abaqus	
Course Code	SIM-en-NVH-A-V30R2023
Available Releases	SIMULIA 2018 , SIMULIA 2019 , SIMULIA 2020 , SIMULIA 2021 , SIMULIA 2022 , SIMULIA 2023 , SIMULIA V6.12
Duration	19.92 hours
Course Material	
Level	Advanced
Audience	This course is recommended for simulation analysts who will perform analyses related to automotive NVH.
Description	Vehicle NVH (Noise, Vibration and Harshness) is typically perceived as a reflection of vehicle quality. As a result, the primary goal of NVH design is to optimize the energy absorption of the vehicle. Large-scale linear dynamics is typically employed in NVH analysis. This course focuses on applying the linear dynamics capabilities in Abaqus to NVH-related simulation. Additional appendix material is included to provide guidance for users translating Nastran models to Abaqus.
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 plugins)

Automotive NVH with Abaqus	
Prerequisites	Introduction to Abaqus or equivalent and experience using Abaqus. Linear Dynamics with Abaqus is also recommended but not required.
Available Online	Yes

Buckling, Postbuckling and Collapse Analysis	
Course Code	SIM-en-BUCK-A-V30R2023
Available Releases	SIMULIA 2018 , SIMULIA 2019 , SIMULIA 2020 , SIMULIA 2021 , SIMULIA 2022 , SIMULIA 2023
Duration	16.17 hours
Course Material	
Level	Advanced
Audience	This course is recommended for simulation analysts who will perform buckling, postbuckling and collapse analysis.
Description	Buckling and postbuckling behavior is critical to the success of certain designs. For example, crashworthiness of an automobile requires that particular vehicle components collapse in ways that maximize energy absorption. On the other hand, successful designs of imperfection-sensitive, thin-walled shell structures, ranging from beverage containers to large pressure vessels, must prevent unintentional buckling. This course blends the theoretical background on such topics as geometric nonlinearity and the Riks method together with examples, guidelines and workshops to illustrate how to simulate buckling and postbuckling behavior.
Objectives	Upon completion of this course you will be able to: - Identify an imperfection-sensitive structure - Extract closely-spaced eigenvalues efficiently - Introduce imperfections into a “perfect” mesh - Use the Riks method effectively - Use damping to control unstable motions
Prerequisites	Introduction to Abaqus or equivalent and experience using Abaqus.

Buckling, Postbuckling and Collapse Analysis	
Available Online	Yes

Composites Modeler for Abaqus/CAE	
Course Code	SIM-en-CMA-A-V30R2023
Available Releases	SIMULIA 2018 , SIMULIA 2019 , SIMULIA 2020 , SIMULIA 2021 , SIMULIA 2022 , SIMULIA 2023 , SIMULIA V6.12
Duration	15.17 hours
Course Material	
Level	Advanced
Audience	This course is recommended for users who are already familiar with the native Abaqus/CAE composites modeling functionality.
Description	This is a two-day course on Composites Modeler for Abaqus/CAE, an add-on product that complements and extends the powerful ply modeling features in Abaqus/CAE. Composites Modeler for Abaqus/CAE provides proven fiber simulation capabilities and advanced model building—all seamlessly integrated within Abaqus/CAE. It allows you to model composite structures in Abaqus/CAE in a way that reflects the composite's manufacturing processes.
Objectives	In this course you will learn: - 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	
Prerequisites	The Analysis of Composite Materials with Abaqus course 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

Connector Elements and Mechanism Analysis with Abaqus	
Course Code	SIM-en-FLEX-A-V30R2023
Available Releases	SIMULIA 2019 , SIMULIA 2020 , SIMULIA 2021 , SIMULIA 2022 , SIMULIA 2023
Duration	16.67 hours
Course Material	
Level	Advanced
Audience	This course is recommended for simulation analysts who will model connections of varying complexity between multiple (rigid or flexible) bodies.
Description	The combination of mechanisms, rigid bodies, and finite elements in Abaqus makes a powerful simulation tool. The mechanism capability expands the ability of Abaqus to model connections among individual bodies in a variety of ways. Connections can be as simple as pins and rigid links or as complicated as nonlinear frictional joints with elasticity and failure. In this course you will explore the variety of connection types available in Abaqus and learn how to define connections that suit your needs.
Objectives	The following topics covered in this course: - Comparison of connectors and MPCs - Basic connector components - Assembled kinematic connections - Local relative displacements and rotations - Connector elasticity - Connector friction - Connector failure - Stops and locks - Actuating components of relative motion - Sensors and actuators

Connector Elements and Mechanism Analysis with Abaqus	
	- Output and postprocessing
Prerequisites	Introduction to Abaqus or equivalent and experience using Abaqus.
Available Online	Yes

Co-simulation with Abaqus and Dymola	
Course Code	SIM-en-DYM-A-V30R2023
Available Releases	SIMULIA 2018 , SIMULIA 2019 , SIMULIA 2020 , SIMULIA 2021 , SIMULIA 2022 , SIMULIA 2023 , SIMULIA V6.12
Duration	5.50 hours
Course Material	
Level	Advanced
Audience	This course is recommended for simulation analysts who need to include logical modeling elements into their simulations.
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. In a typical workflow, sensor data computed in Abaqus (the physical system) are passed to Dymola (the controller), which in turn computes the needed actuation to drive the Abaqus model to a desired state. The powerful logical modeling features in Dymola cover a wide variety of engineering fields, such as electromechanics, control systems, hydraulics, penumatics, etc. This course illustrates coupling Dymola and Abaqus to develop a very versatile logical-physical modeling capability.
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 for co-simulation - Run a co-simulation between Abaqus and Dymola models

Co-simulation with Abaqus and Dymola	
	- Review the co-simulation results
Prerequisites	Introduction to Abaqus or equivalent. Experience using Abaqus and a basic understanding of control systems. Some familiarity with Dymola usage is helpful but not required.
Available Online	Yes

Crashworthiness Analysis with Abaqus	
Course Code	SIM-en-CRASH-A-V30R2023
Available Releases	SIMULIA 2018 , SIMULIA 2019 , SIMULIA 2020 , SIMULIA 2021 , SIMULIA 2022 , SIMULIA 2023 , SIMULIA V6.12
Duration	19.17 hours
Course Material	
Level	Advanced
Audience	This course is recommended for simulation analysts 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. The course is ideal for users who are already familiar with other crash codes and would like to transition to Abaqus. The course does not cover the use of preprocessors and introduces you to the analysis capabilities of Abaqus using the keyword interface. The course uses examples derived from actual industrial applications to reinforce the concepts and issues discussed in the lectures.
Objectives	This course covers the following topics: - 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.

Crashworthiness Analysis with Abaqus	
Available Online	Yes

CZone for Abaqus	
Course Code	SIM-en-CZA-A-V30R2023
Available Releases	SIMULIA 2018 , SIMULIA 2019 , SIMULIA 2020 , SIMULIA 2021 , SIMULIA 2022 , SIMULIA 2023 , SIMULIA V6.12
Duration	4.67 hours
Course Material	
Level	Advanced
Audience	This course is recommended for experienced users of Abaqus/Explicit who will perform crush analysis of composite structures.
Description	This half-day course is an introduction to CZone for Abaqus (CZA), an add-on capability to Abaqus/Explicit that provides access to a state-of-the-art methodology for crush simulation. Based on CZone technology from Engenuity Ltd., CZone for Abaqus provides for inclusion of material crush behavior in Abaqus/ Explicit simulations of composite structures subjected to impact.
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 postprocess CZA analysis results
Prerequisites	The Abaqus/Explicit: Advanced Topics and Analysis of Composite Materials with Abaqus courses are recommended as prerequisites. Experience using Abaqus/Explicit is also recommended.

CZone for Abaqus	
Available Online	Yes

Electromagnetic Analysis with Abaqus	
Course Code	SIM-en-EMAG-A-V30R2023
Available Releases	SIMULIA 2018 , SIMULIA 2019 , SIMULIA 2020 , SIMULIA 2021 , SIMULIA 2022 , SIMULIA 2023 , SIMULIA V6.12
Duration	9.17 hours
Course Material	
Level	Advanced
Audience	This course is recommended for simulation analysts who will perform electromagnetic simulation with Abaqus.
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 do the following with Abaqus: - Set up and create electromagnetic models - Perform low frequency eddy current analyses - Perform transient eddy current analyses - Perform magnetostatic analyses
Prerequisites	Introduction to Abaqus or equivalent and experience using Abaqus.
Available Online	Yes

Element Selection in Abaqus	
Course Code	SIM-en-ELEMC-A-V30R2023
Available Releases	SIMULIA 2018 , SIMULIA 2019 , SIMULIA 2020 , SIMULIA 2021 , SIMULIA 2022 , SIMULIA 2023 , SIMULIA V6.12
Duration	13.67 hours
Course Material	
Level	Advanced
Audience	This course is recommended for simulation analysts who will perform advanced nonlinear analysis, regardless of industry or application.
Description	Choosing an element is one of the most fundamental questions that users must answer as they build a finite element model. Many issues should be considered when selecting an element, including: Is there contact in the model? Is the material behavior fully or nearly incompressible? Will the element bend during the analysis? Is the structure thick or thin? Will the mesh become severely distorted? What results are needed from the analysis? Is the analysis a static or dynamic simulation? This course provides an overview of the different element types available in Abaqus for stress analyses. Differences between the Abaqus/Standard and Abaqus/Explicit element libraries are considered. Application-specific recommendations for choosing an element type are provided. The pitfalls and symptoms of choosing incorrect element types are also discussed. Examples and workshops are used to illustrate element behavior and the consequences of choosing incorrect element types for a given problem
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

Element Selection in Abaqus	
	- 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	Introduction to Abaqus or equivalent and experience using Abaqus.
Available Online	Yes

Fatigue of Welds in fe-safe (FWFS)	
Course Code	SIM-en-FWFS-A-V30R2023
Available Releases	SIMULIA 2019 , SIMULIA 2020 , SIMULIA 2021 , SIMULIA 2022 , SIMULIA 2023
Duration	5.58 hours
Course Material	
Level	Advanced
Audience	Simulation Analysts
Description	This course covers simulating the number of cycles till fatigue failures at welds in welded structures. The methods are based on predicting a crack through a certain thickness, in welded structures at the welds (toe, root, throat, etc.; failure mechanisms in welds can be included). Both classification methods (such as BS5400 and BS7608) and structural stress methods (such as the Verity® module in fe-safe) are included. Examples are provided using FEA output models run in Abaqus, ANSYS or Nastran. Not included is temperature-dependent weld fatigue, or weld processes such as friction stir welding or similar. Non-structural joining such as solder is not covered, nor is nonmetallic joining such as adhesive bonding.
Objectives	Upon completion of this course you will be able to: - Understand modeling requirements for line welds and spot welds - Set up FEA models for weld fatigue and import results into fe-safe - Use BS5400 Weld Finite Life algorithm in fe-safe - Define weld lines for structural stress calculations - Use the Verity module in fe-safe based on structural stress calculations - Select a material for fatigue analysis using structural stress methods

Fatigue of Welds in fe-safe (FWFS)	
	- Run weld methods and/or base metal fatigue simultaneously - Postprocess weld fatigue in fe-safe and using a postprocessor
Prerequisites	Introduction to fe-safe
Available Online	Yes

Fitness-for-Service Analysis with Abaqus	
Course Code	SIM-en-FFSA-F-V30R2023
Available Releases	SIMULIA 2018 , SIMULIA 2019 , SIMULIA 2020 , SIMULIA 2021 , SIMULIA 2022 , SIMULIA 2023
Duration	14.67 hours
Course Material	
Level	Fundamental
Audience	This course is recommended for Fixed-Equipment Reliability engineers, Fitness-for-Service engineers, Plant process engineers, Plant-equipment inspectors, Plant managers.
Description	Pressure vessels and piping equipment are periodically assessed based on guidelines prescribed by documents such as ASME/API-579 Fitness-for-Service (FFS). Finite element based Level – 3 assessments are often utilized for assessing equipment with complex geometries and loading conditions. This course discusses methods for modeling common pressure vessels such as distillation towers, storage vessels, etc. using Abaqus/CAE. Methods for application and verification of loads such as weight of contents, internal pressure, etc. using Abaqus/ Standard, as required for Level-3 FFS assessments are also discussed. Procedures for analyzing metal loss using the finite element method by mapping thickness readings from scans are also considered. The following products are covered by this course: Abaqus/CAE, Abaqus/Standard. The course is divided into lectures and workshops. The course's workshops are integral to the training. They are designed to reinforce concepts presented during the lectures. They are intended to provide users with the experience of running and trouble-shooting actual Abaqus analyses.
Objectives	The following topics are covered in this course:

Fitness-for-Service Analysis with Abaqus	
	- Linear and Nonlinear finite element analysis procedures - Modeling pressure vessel geometries - Modeling loads and boundary conditions - Fitness-for-Service assessment procedures - FEA based Level – 3 Assessment
Prerequisites	None
Available Online	Yes

FSI Simulation with Abaqus and Third-Party CFD Codes	
Course Code	SIM-en-FSI-A-V30R2020
Available Releases	SIMULIA 2018 , SIMULIA 2019 , SIMULIA 2020 , SIMULIA V6.12
Duration	8 hours
Course Material	
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	
Course Code	SIM-en-GUIC-A-V30R2023
Available Releases	SIMULIA 2018 , SIMULIA 2019 , SIMULIA 2020 , SIMULIA 2021 , SIMULIA 2022 , SIMULIA 2023 , SIMULIA V6.12
Duration	16.92 hours
Course Material	
Level	Advanced
Audience	This course is recommended for users interested in modifying and extending the capabilities of Abaqus by customizing their Abaqus interface.
Description	Increasingly, Abaqus is being applied to more routine, repeatable workflows and analysis procedures. These workflows can be captured using specialized Abaqus Process Automation tools. One of the more commonly used Process Automation tools is the Abaqus GUI Toolkit. The Abaqus GUI Toolkit provides programming routines that allow you to change the Abaqus graphical user interface (GUI) and build customized applications. Such applications can enable you to capture proven Abaqus-centric workflows and methods for deployment to a wider range of users and generate Abaqus solutions more efficiently and reliably. This course covers basic syntax and the fundamentals of the Abaqus GUI Toolkit through a combination of lectures, examples and workshops. The hands-on workshops are an integral component of learning about the Abaqus GUI Toolkit and represent a significant portion of the course experience. The workshops focus on distinct aspects of the GUI Toolkit and build upon each other in order to create a complete stand-alone application.
Objectives	Upon completion of this course you will be able to: - Build dialogs and issue commands from the GUI

GUI Customization with Abaqus	
	- Create and modify GUI modules and toolsets - Create custom applications
Prerequisites	This course assumes prior knowledge of the Python programming language and the Abaqus kernel commands. Thus, students must attend the Introduction to Abaqus Scripting course 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	
Course Code	SIM-en-HEAT-A-V30R2023
Available Releases	SIMULIA 2018 , SIMULIA 2019 , SIMULIA 2020 , SIMULIA 2021 , SIMULIA 2022 , SIMULIA 2023 , SIMULIA V6.12
Duration	13 hours
Course Material	
Level	Advanced
Audience	This course is recommended for simulation analysts whose work involves assessing the thermal response of structures and/or the thermal influence on mechanical response.
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. This course introduces you to the heat transfer and thermal-stress capabilities available within Abaqus. Practical examples and workshops are used to illustrate these capabilities.
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 sequentially-coupled, fully-coupled and adiabatic thermal-stress analyses - Model contact in heat transfer problems

Heat Transfer and Thermal-Stress Analysis with Abaqus	
Prerequisites	Introduction to Abaqus or equivalent and experience using Abaqus.
Available Online	Yes

Introduction to Abaqus	
Course Code	SIM-en-ABI-F-V30R2023
Available Releases	SIMULIA 2018 , SIMULIA 2019 , SIMULIA 2020 , SIMULIA 2021 , SIMULIA 2022 , SIMULIA 2023 , SIMULIA V6.12
Duration	33.67 hours
Course Material	
Level	Fundamental
Audience	This course is recommended for new Abaqus users who will primarily use Abaqus/CAE to create their models.
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. The following products are covered: Abaqus/CAE, Abaqus/Standard and Abaqus/ Explicit.
Objectives	The course covers the following topics: - Linear and nonlinear structural analysis - Static, dynamic and heat transfer analysis - Material models: linear elasticity, hyperelasticity and metal plasticity - Loads and constraints - Modeling contact - Selecting the appropriate elements for your problem - Feature-based modeling, parts and assemblies - Working with CAD geometry and imported meshes - Mesh generation techniques - Creating, submitting and monitoring analysis jobs - Viewing simulation results - Restarting an analysis The course is divided into lectures, demonstrations and workshops. The

Introduction to Abaqus	
	course's workshops are integral to the training. They are designed to reinforce concepts presented during the lectures and demonstrations. They are intended to provide users with the experience of running and trouble-shooting actual Abaqus analyses.
Prerequisites	No previous knowledge of Abaqus is required, but some basic knowledge of finite elements, interactive modeling and continuum mechanics is desirable.
Available Online	Yes

Introduction to Abaqus/CAE	
Course Code	SIM-en-ICAE-F-V30R2023
Available Releases	SIMULIA 2018 , SIMULIA 2019 , SIMULIA 2020 , SIMULIA 2021 , SIMULIA 2022 , SIMULIA 2023 , SIMULIA V6.12
Duration	15.67 hours
Course Material	
Level	Fundamental
Audience	This course is recommended for new Abaqus/CAE users.
Description	This introductory course introduces you to Abaqus/ CAE, the interactive interface for Abaqus products. 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. - Submit and monitor analysis jobs. - View and evaluate simulation results.
Prerequisites	Some familiarity with interactive preprocessors is helpful but not required.
Available Online	Yes

Introduction to Abaqus/Standard and Abaqus/Explicit	
Course Code	SIM-en-IABA-F-V30R2023
Available Releases	SIMULIA 2018 , SIMULIA 2019 , SIMULIA 2020 , SIMULIA 2021 , SIMULIA 2022 , SIMULIA 2023 , SIMULIA V6.12
Duration	17.17 hours
Course Material	
Level	Fundamental
Audience	This course is recommended for new Abaqus users who will primarily use third-party preprocessors and whose work will require routine editing of input files.
Description	This introductory course introduces you to the analysis capabilities of Abaqus using the Abaqus keywords interface. It is the ideal way for those who will primarily use third-party preprocessors to obtain a working knowledge of both Abaqus/Standard and Abaqus/Explicit to solve linear and nonlinear problems. The following products are covered: Abaqus/Standard, Abaqus/Explicit and Abaqus/Viewer. If interested in the interactive interface of Abaqus, consider the Introduction to Abaqus course instead.
Objectives	The course covers the following topics: - Fundamental modeling techniques and syntax - Linear and nonlinear statics - Selection of the appropriate element for your problem - Adaptive load incrementation and convergence criteria - Interpretation of messages issued by Abaqus - Geometric, material and contact-induced nonlinearity - Linear elasticity and metal plasticity - Restarting of analyses

Introduction to Abaqus/Standard and Abaqus/Explicit	
	- Appropriate modeling techniques for contact problems - Eigenfrequency extraction - Linear and nonlinear dynamics - Model transfer between Abaqus/Explicit and Abaqus/Standard The course's workshops are an integral part of the training. They are designed to reinforce the concepts presented during the lectures and to provide users with the experience of running and trouble-shooting actual Abaqus analyses. The workshops also provide instruction on using Abaqus postprocessors for results evaluation.
Prerequisites	No previous knowledge of Abaqus is required, but some basic knowledge of finite elements and continuum mechanics is desirable.
Available Online	Yes

Introduction to Abaqus Scripting	
Course Code	SIM-en-ISRPT-F-V30R2023
Available Releases	SIMULIA 2018 , SIMULIA 2019 , SIMULIA 2020 , SIMULIA 2021 , SIMULIA 2022 , SIMULIA 2023 , SIMULIA V6.12
Duration	16.67 hours
Course Material	
Level	Fundamental
Audience	We believe every Abaqus user should be familiar with the ASI. Productivity gains can be realized at many levels; from simple scripts that automate tedious tasks to large applications with custom user interfaces. This course is recommended for all Abaqus users who wish to improve their productivity using scripting.
Description	Abaqus makes extensive use of Python, a powerful, object-oriented scripting language that is used widely by organizations throughout the world. Python has been embedded within the Abaqus software products. The language has been extended to include a rich set of commands that are well suited for the daily tasks of a finite element analyst. These extensions are referred to as the Abaqus Scripting Interface (ASI). The Abaqus Scripting Interface may be used by the finite element analyst at many different levels. Scripts can be written as stand-alone utilities, or can be written to integrate the Abaqus products with other codes. At a basic level, scripts may be used to automate repetitive tasks such as the creation of results plots from a collection of output files. With some experience, users may actually extend the functionality of the Abaqus products. Advanced users may work with SIMULIA affiliates to customize the graphical user interface of the Abaqus interactive products (Abaqus/CAE and Abaqus/Viewer). This course covers basic usage of the Abaqus Scripting Interface and Python's syntax. It

Introduction to Abaqus Scripting	
	includes numerous hands-on exercises for the student to learn to automate tasks that are common to most analysts.
Objectives	The goals of the course are to allow the student to: - Develop a high-level understanding of the Abaqus scripting capabilities. - Understand the technical details of Python and the Abaqus Scripting Interface. - Understand the strengths and weaknesses of Abaqus scripting.
Prerequisites	In order for the training to be effective, all students should have some basic familiarity with the Abaqus products including Abaqus/Viewer and Abaqus/ Standard or Abaqus/Explicit. Familiarity with Abaqus/ CAE is very helpful. Students should also have some experience using at least one computer programming language, text editing, and should be proficient with basic operating system tasks such as file copying/ deleting, creating/modifying environment variables, etc.
Available Online	Yes

Introduction to fe-safe	
Course Code	SIM-en-IFES-F-V30R2023
Available Releases	SIMULIA 2018 , SIMULIA 2019 , SIMULIA 2020 , SIMULIA 2021 , SIMULIA 2022 , SIMULIA 2023
Duration	10.75 hours
Course Material	
Level	Fundamental
Audience	Simulation Analysts
Description	Durability is a requirement for product creation in many industries, because cycling loads can cause damage much sooner than a static load. This course introduces fatigue life calculations in fe-safe and gives you an understanding of how to configure a fatigue analysis based on FEA solutions. These life results can be used to iterate on a design and improve the overall durability of the design. The course is divided into lectures and workshops. The course's workshops are integral to the training. They are designed to reinforce concepts presented during the lectures. They are intended to provide users with the experience of running and trouble-shooting actual fatigue analyses.
Objectives	Upon completion of this course you will be able to set up and run various fatigue analyses using fe-safe, including: - Import FEA solutions into fe-safe - Select a material for fatigue analysis - Define fatigue loadings - Run room temperature fatigue analysis in fe-safe - Understand the most commonly used fatigue algorithms - Use factor of strength to evaluate a target life - Understand the fatigue life results - Request additional exports and outputs

Introduction to fe-safe	
Prerequisites	None
Available Online	Yes

Introduction to fe-safe/Rubber	
Course Code	SIM-en-IFSR-F-V30R2022
Available Releases	SIMULIA 2018 , SIMULIA 2019 , SIMULIA 2020 , SIMULIA 2021 , SIMULIA 2022
Duration	16 hours
Course Material	
Level	Fundamental
Audience	Simuation Analyst: This course is recommended for engineers with experience using an FEA solver and the fe-safe GUI. Some overview of the fe-safe/Rubber GUI is given, but users are expected to know how to use the Abaqus/CAE and Abaqus/Viewer interfaces. Suggested prerequisites: Introduction to Abaqus and Introduction to fe-safe.
Description	This 2-day course provides background information and hands-on experience for calculating fatigue of elastomers using Abaqus and fe-safe/Rubber
Objectives	FEA Modeling for fe-safe/Rubber in Abaqus/CAE - Using the fe-safe/Rubber Interface - Stress-strain relationship for rubber materials in Abaqus - Theory of fatigue crack growth under relaxing and non-relaxing loads - Overview of material calibration and how to enter material properties in fe-safe - Calibrating the crack precursor size for fe-safe/ Rubber - Variable amplitude loading and multiple block loading for fe-safe/Rubber - Postprocessing exports from fe-safe/Rubber (spreadsheet and Abaqus/Viewer) - Using fe-safe/Rubber output in the stand-alone Endurica damage sphere viewer

Introduction to fe-safe/Rubber	
	- Industry examples
Prerequisites	- Introduction to Abaqus - Introduction to fe-safe
Available Online	Yes

Introduction to Isight	
Course Code	SIM-en-ISGT-F-V30R2023
Available Releases	SIMULIA 2018 , SIMULIA 2019 , SIMULIA 2020 , SIMULIA 2021 , SIMULIA 2022 , SIMULIA 2023
Duration	14.17 hours
Course Material	
Level	Fundamental
Audience	The course is recommended for new Isight users and anyone else interested in learning more about Isight, including mechanical designers, analysts and methods developers.
Description	This course provides a practical introduction to Isight in which you will learn about process integration and parametric design optimization using Isight. The course includes many hands-on workshops and practical examples.
Objectives	
Prerequisites	None
Available Online	Yes

Introduction to Tosca Structure	
Course Code	SIM-en-TOSCST-F-V30R2022
Available Releases	SIMULIA 2018 , SIMULIA 2019 , SIMULIA 2020 , SIMULIA 2021 , SIMULIA 2022
Duration	15.42 hours
Course Material	
Level	Fundamental
Audience	Simulation Analysts
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

Isight Component Development	
Course Code	SIM-en-ISCD-A-V30R2023
Available Releases	SIMULIA 2018 , SIMULIA 2019 , SIMULIA 2020 , SIMULIA 2021 , SIMULIA 2022 , SIMULIA 2023
Duration	26.33 hours
Course Material	
Level	Advanced
Audience	Simulation Analysts
Description	Isight is a powerful tool for creating flexible simulation workflows using an extensive library of built-in components. However, it is possible to extend this library by developing custom components which can provide interfaces to third-party simulation codes and/ or extend existing components via custom plug-ins using the power of the Java development language. 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	The topics discussed include the following: - Isight component architecture and introduction to the Isight SKD - Building and testing an Isight component with a custom User Interface - Interfacing with third-party simulation codes written in other languages such as Fortran - Extending the behavior of existing Isight library components - Introduction to the Isight developers plug-in and debugging features using Eclipse IDE

Isight Component Development	
	- Build a custom DOE (Design of Experiments) method plug-in
Prerequisites	The course is recommended for simulation analysts and methods developers who have experience with Isight. Students should be familiar with software development using the Java language.
Available Online	Yes

Linear Dynamics with Abaqus	
Course Code	SIM-en-LNDYN-A-V30R2023
Available Releases	SIMULIA 2018 , SIMULIA 2019 , SIMULIA 2020 , SIMULIA 2021 , SIMULIA 2022 , SIMULIA 2023 , SIMULIA V6.12
Duration	16.33 hours
Course Material	
Level	Advanced
Audience	This course is recommended for simulation analysts who will perform linear dynamic analysis.
Description	This course introduces the user to the algorithms and methods used to study linear dynamic problems with Abaqus/Standard. The workshops reinforce the fundamental concepts presented in the lectures.
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	Introduction to Abaqus or equivalent and experience using Abaqus.
Available Online	Yes

Metal Forming with Abaqus	
Course Code	SIM-en-METF-A-V30R2023
Available Releases	SIMULIA 2018 , SIMULIA 2019 , SIMULIA 2020 , SIMULIA 2021 , SIMULIA 2022 , SIMULIA 2023 , SIMULIA V6.12
Duration	20.42 hours
Course Material	
Level	Advanced
Audience	This course is recommended for simulation analysts who will perform metal forming analysis.
Description	Metal forming processes are highly nonlinear because they involve geometric, material and contact nonlinearities. Therefore, simulating these processes numerically can be a difficult task. However, numerical simulations of forming processes present advantages that outweigh the difficulties. Numerical simulation can reduce both the cost and length of a product development cycle by identifying potential forming problems prior to tooling fabrication and reducing the time and cost associated with tooling rework. Numerical simulation can also improve the quality of the part being manufactured through testing to ensure that the manufacturing processes appropriately account for springback and stretching of the parts. Using lectures and hands-on workshops, this course provides practical modeling skills and techniques for simulating forming processes.
Objectives	In this course you will learn practical modeling skills and techniques for: - Stamping - Hydroforming - Punch stretching - Forging - Rolling

Metal Forming with Abaqus	
	- Drawing - Superplastic forming
Prerequisites	Introduction to Abaqus or equivalent and experience using Abaqus.
Available Online	Yes

Metal Inelasticity in Abaqus	
Course Code	SIM-en-METAL-A-V30R2023
Available Releases	SIMULIA 2018 , SIMULIA 2019 , SIMULIA 2020 , SIMULIA 2021 , SIMULIA 2022 , SIMULIA 2023 , SIMULIA V6.12
Duration	12.17 hours
Course Material	
Level	Advanced
Audience	This course is recommended for simulation analysts whose work involves evaluating the inelastic response of metals.
Description	Properly modeling the inelastic behavior of materials is very important when evaluating the performance of a design in critical loading situations. This course provides a brief overview of the inelastic behavior observed in metals and the basic concepts of plasticity theory. The course shows Abaqus users how to model various forms of metal plasticity using a combination of examples and workshops to demonstrate the material models and the type of experimental data necessary to calibrate them. The assumptions and limitations of the various plasticity and creep models are also discussed.
Objectives	Upon completion of this course you will be able to model: - 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

Metal Inelasticity in Abaqus	
	- Creep behavior in metals
Prerequisites	Introduction to Abaqus or equivalent and experience using Abaqus.
Available Online	Yes

Modeling Contact and Resolving Convergence Issues with Abaqus	
Course Code	SIM-en-MCRC-A-V30R2023
Available Releases	SIMULIA 2018 , SIMULIA 2019 , SIMULIA 2020 , SIMULIA 2021 , SIMULIA 2022 , SIMULIA 2023
Duration	22.17 hours
Course Material	
Level	Advanced
Audience	This course is recommended for simulation analysts who will perform advanced nonlinear analysis, regardless of industry or application.
Description	
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 - Define contact interactions - Define appropriate surfaces (rigid or deformable) - Model frictional contact - Model large sliding between deformable bodies - Resolve overclosures in interference fit problems
Prerequisites	Introduction to Abaqus or equivalent and experience using Abaqus.
Available Online	Yes

Modeling Contact with Abaqus/Standard	
Course Code	SIM-en-CONT-A-V30R2023
Available Releases	SIMULIA 2018 , SIMULIA 2019 , SIMULIA 2020 , SIMULIA 2021 , SIMULIA 2022 , SIMULIA 2023 , SIMULIA V6.12
Duration	18.17 hours
Course Material	
Level	Advanced
Audience	This course is recommended for simulation analysts who will perform advanced nonlinear analysis, regardless of industry or application.
Description	Understanding the interaction between bodies is essential for solving many engineering problems. Manufacturing processes, gears, bearings, seals and dynamic impact events all involve contact. This course covers many techniques and guidelines for solving challenging contact problems. 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 - Avoid overconstraining the model - Avoid rigid body motions and unstable motions - Use pre-tension sections to simulate assembly loads

Modeling Contact with Abaqus/Standard	
Prerequisites	Introduction to Abaqus or equivalent and experience using Abaqus.
Available Online	Yes

Modeling Extreme Deformation and Fluid Flow with Abaqus	
Course Code	SIM-en-FLOW-A-V30R2023
Available Releases	SIMULIA 2018 , SIMULIA 2019 , SIMULIA 2020 , SIMULIA 2021 , SIMULIA 2022 , SIMULIA 2023 , SIMULIA V6.12
Duration	11.67 hours
Course Material	
Level	Advanced
Audience	This course is recommended for simulation analysts whose work involves modeling extreme deformation and fluid flow.
Description	Abaqus includes several advanced techniques for modeling extreme deformation and fluid flow. The pure Eulerian analysis capability in Abaqus/Explicit allows for effective modeling of fluid flows and extremely large deformations in solids. Coupling the power of this capability with the traditional Lagrangian approach makes it possible to simulate complicated multiphysics phenomena such as fluid-structure interactions where highly deformable materials interact with relatively stiff bodies. The need for modeling high-velocity impacts, extremely violent fluid flows, and material phase changes requires the use of Smoothed Particle Hydrodynamics, a meshless Lagrangian method capable of solving these challenging problems efficiently. This course 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.

Modeling Extreme Deformation and Fluid Flow with Abaqus	
Objectives	Upon completion of this course you will be able to: Coupled Eulerian-Lagrangian (CEL) method - Create Eulerian meshes and define the initial material location within an Eulerian mesh - Apply initial conditions, boundary conditions and loads to materials in the Eulerian domain - Use general contact to model Eulerian-Lagrangian interactions - Model fluids using the Equation-of-State material model - Visualize material boundaries within the Eulerian domain Smoothed Particle Hydrodynamic (SPH) method - 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 and element-based or analytical surfaces - Understand the differences between the CEL and SPH approaches
Prerequisites	Introduction to Abaqus or equivalent and experience using Abaqus.
Available Online	Yes

Modeling Fracture and Failure with Abaqus	
Course Code	SIM-en-FRAC-A-V30R2023
Available Releases	SIMULIA 2018 , SIMULIA 2019 , SIMULIA 2020 , SIMULIA 2021 , SIMULIA 2022 , SIMULIA 2023 , SIMULIA V6.12
Duration	20.17 hours
Course Material	
Level	Advanced
Audience	This course is recommended for simulation analysts who will perform fracture and failure studies.
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 course provides a detailed discussion of these capabilities.
Objectives	Upon completion of this course you will be able to: - Use proper modeling techniques to capture crack-tip singularities in fracture mechanics problems - Use proper modeling techniques for finite-strain (nonlinear) 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	Introduction to Abaqus or equivalent and experience using Abaqus.

Modeling Fracture and Failure with Abaqus	
Available Online	Yes

Modeling Rubber and Viscoelasticity with Abaqus	
Course Code	SIM-en-MRUB-A-V30R2023
Available Releases	SIMULIA 2018 , SIMULIA 2019 , SIMULIA 2020 , SIMULIA 2021 , SIMULIA 2022 , SIMULIA 2023 , SIMULIA V6.12
Duration	12.67 hours
Course Material	
Level	Advanced
Audience	This course is recommended for simulation analysts who will perform analysis of rubber components.
Description	Rubber and resilient foam are widely used for a variety of applications, such as seals and gaskets, shock mounts, vibration isolators and tires. The mechanical and chemical properties of these materials allow them to act as excellent seals against moisture, pressure and heat. They also have excellent energy absorption and dissipation capabilities. This course provides a brief overview of finite deformations and the material models available in Abaqus which are 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

Modeling Rubber and Viscoelasticity with Abaqus	
Prerequisites	Introduction to Abaqus or equivalent and experience using Abaqus.
Available Online	Yes

Modeling Stents Using Abaqus	
Course Code	SIM-en-STENT-A-V30R2023
Available Releases	SIMULIA 2018 , SIMULIA 2019 , SIMULIA 2020 , SIMULIA 2021 , SIMULIA 2022 , SIMULIA 2023 , SIMULIA V6.12
Duration	10.92 hours
Course Material	
Level	Advanced
Audience	This course is recommended for simulation analysts responsible for the design of medical devices.
Description	The simulation of stent behavior reveals detailed stress-strain distributions that are important in predicting device fatigue life. The complex geometry and material behavior of stents result in highly nonlinear and challenging analyses. 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. This course will also prepare attendees to take the next step to simulate stent behavior through in silico representations of the vasculature and using the Living Heart Human Models.
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

Modeling Stents Using Abaqus	
	- Define contact and constraints - Postprocess stent analyses - Perform fatigue evaluation
Prerequisites	This course is recommended for engineers with experience using Abaqus. Some understanding of Abaqus/CAE is helpful but is not required.
Available Online	Yes

Obtaining a Converged Solution with Abaqus	
Course Code	SIM-en-CONV-A-V30R2023
Available Releases	SIMULIA 2018 , SIMULIA 2019 , SIMULIA 2020 , SIMULIA 2021 , SIMULIA 2022 , SIMULIA 2023 , SIMULIA V6.12
Duration	15.17 hours
Course Material	
Level	Advanced
Audience	This course is recommended for simulation analysts who will perform advanced nonlinear analysis, regardless of industry or application.
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. This course provides an in-depth discussion on solving nonlinear problems in Abaqus/Standard and addressing the most common convergence-related issues. 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	Introduction to Abaqus or equivalent and experience using Abaqus.

Obtaining a Converged Solution with Abaqus	
Available Online	Yes

Optimizing Engineering Methods with Isight	
Course Code	SIM-en-ISOM-A-V30R2023
Available Releases	SIMULIA 2018 , SIMULIA 2019 , SIMULIA 2020 , SIMULIA 2021 , SIMULIA 2022 , SIMULIA 2023
Duration	12.17 hours
Course Material	
Level	Advanced
Audience	Simulation Analysts, Scientists
Description	This course provides a brief overview of Isight and optimization before discussing nonlinear optimization theories and applications. Topics such as design space searching, multi-objective optimization, optimization strategy, and multidisciplinary optimization are covered. Attendees will learn key differences between the optimization algorithms offered in Isight, how to choose the preferred method based on the problem, how to remedy issues with run-time performance, and other topics relevant to improving the usage and value of Isight for real engineering optimization problems.
Objectives	The topics discussed include the following: - Design Space Exploration for Optimization problems - Optimization techniques (Gradient Based, Pattern Methods, Exploratory Methods) - Multi Objective Optimization - Nested Exploration and Adaptive DOE - Exploration techniques (Pointer and Pointer 2) - Optimization technique selection strategy
Prerequisites	Introduction to Isight
Available Online	Yes

Structural-Acoustic Analysis Using Abaqus	
Course Code	SIM-en-ACOU-A-V30R2023
Available Releases	SIMULIA 2018 , SIMULIA 2019 , SIMULIA 2020 , SIMULIA 2021 , SIMULIA 2022 , SIMULIA 2023 , SIMULIA V6.12
Duration	13.92 hours
Course Material	
Level	Advanced
Audience	This course is recommended for simulation analysts who will perform acoustic and structural-acoustic analysis.
Description	Abaqus includes a number of capabilities in the area of structural-acoustic analysis. In addition to pure acoustic analysis features, Abaqus includes the capability to couple nonlinear structural analyses with linear acoustic analyses using several different methods. This seminar covers the fundamentals of acoustics phenomena and then shows how to use Abaqus to solve a wide range of acoustics problems.
Objectives	After completing this course you will gain understanding of: - Basics of acoustic phenomena - Radiation boundary conditions - Element selection - Mesh size and mesh density effects for different analysis procedures - Pure acoustics analysis - Coupled structural-acoustic analysis - Scattering and shock analysis - Analysis procedures available for acoustic analysis - Output and postprocessing

Structural-Acoustic Analysis Using Abaqus	
Prerequisites	Introduction to Abaqus or equivalent and experience using Abaqus. Some understanding of acoustics is helpful but not required.
Available Online	Yes

Substructures and Submodeling with Abaqus	
Course Code	SIM-en-SUPSUB-A-V30R2023
Available Releases	SIMULIA 2018 , SIMULIA 2019 , SIMULIA 2020 , SIMULIA 2021 , SIMULIA 2022 , SIMULIA 2023 , SIMULIA V6.12
Duration	15.17 hours
Course Material	
Level	Advanced
Audience	This course is recommended for simulation analysts who require specialized techniques to study large problems.
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. Substructures are useful to break a large problem into several smaller components. Submodeling allows the analysts to successively focus in on the area of interest.
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	Introduction to Abaqus or equivalent and experience using Abaqus.

Substructures and Submodeling with Abaqus	
Available Online	Yes

Tire Analysis with Abaqus: Advanced Topics	
Course Code	SIM-en-TIRE2-A-V30R2023
Available Releases	SIMULIA 2018 , SIMULIA 2019 , SIMULIA 2020 , SIMULIA 2021 , SIMULIA 2022 , SIMULIA 2023 , SIMULIA V6.12
Duration	8.17 hours
Course Material	
Level	Advanced
Audience	This course is recommended for tire analysts.
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 course 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 - Transient analysis using Abaqus/Explicit - Hydroplaning simulation using Coupled Eulerian-Lagrangian technique - Efficient steady-state dynamics analysis - Substructuring and submodeling
Prerequisites	Tire Analysis with Abaqus: Fundamentals
Available Online	Yes

Tire Analysis with Abaqus: Fundamentals	
Course Code	SIM-en-TIRE-F-V30R2023
Available Releases	SIMULIA 2018 , SIMULIA 2019 , SIMULIA 2020 , SIMULIA 2021 , SIMULIA 2022 , SIMULIA 2023 , SIMULIA V6.12
Duration	12.67 hours
Course Material	
Level	Fundamental
Audience	This course is recommended for tire analysts.
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 course 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	Introduction to Abaqus or equivalent and experience using Abaqus.
Available Online	Yes

Uncertainty Quantification with Isight	
Course Code	SIM-en-ISUQ-A-V30R2023
Available Releases	SIMULIA 2020 , SIMULIA 2021 , SIMULIA 2022 , SIMULIA 2023
Duration	8.25 hours
Course Material	
Level	Advanced
Audience	Simulation Analysts, Design Engineers, Quality Engineers, Manufacturing Engineers, Reliability Engineers, Students and anyone interested in performing stochastic analysis
Description	This course introduces Isight users to methods dealing with statistical behavior as a result of variability in the system. It motivates why uncertainty quantification (UQ) analysis is important, presents concepts and methods in Isight to do UQ analysis, and shows how to use Isight's open architecture to integrate user-developed algorithms into components as plug-ins.
Objectives	Upon completion of this course you will be able to: - Use various Isight components to perform stochastic analysis - Understand concepts used in Taguchi, Reliability and Six Sigma methods
Prerequisites	Introduction to Isight
Available Online	Yes

Writing User Subroutines with Abaqus	
Course Code	SIM-en-SUBR-A-V30R2023
Available Releases	SIMULIA 2018 , SIMULIA 2019 , SIMULIA 2020 , SIMULIA 2021 , SIMULIA 2022 , SIMULIA 2023 , SIMULIA V6.12
Duration	12.92 hours
Course Material	
Level	Advanced
Audience	This course is recommended for engineers who need to adapt the capabilities of the Abaqus solvers to their particular needs.
Description	User subroutines in Abaqus allow the program to be customized for particular applications. This course explains when to consider the use of user subroutines and how to approach their development. Detailed descriptions are given of the data required for selected subroutines, the additional statements to be included and the variables that are available within the routines. Particular attention is paid to highlighting good practice in user subroutine development. Examples of various user subroutines are used to illustrate the points made in the lectures.
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 and UHYPER_STRETCH for modeling hyperelastic materials - UMAT and VUMAT for allowing constitutive models to be added to the program

Writing User Subroutines with Abaqus	
	- UEL and VUEL for allowing the creation of user-defined elements
Prerequisites	User subroutines are a very advanced feature of Abaqus. Thus extensive experience using Abaqus is strongly recommended. In addition, a strong working knowledge of the finite element method and programming in either FORTRAN or C is desirable.
Available Online	Yes

IF WE™