Course Catalog

SIMULIA 2018 / 2019

12 March 2019

3DEXPERIENCE®

Learning Experience | Course Catalog

Learning Experience | Course Catalog

SIMULIA

Learning Experience | Course Catalog

Learning Experience | Course Catalog

Learning Experience | Course Catalog

SIMULIA
Abaqus

Learning Experience | Course Catalog

Abaqus/CAE: Geometry Import and Meshing	
Course Code	SIM-en-CAGIM-A-V30R2019
Available Releases	SIMULIA 2018 , SIMULIA 2019
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

Course Code	SIM-en-ADXP-A-V30R2019
Available Releases	SIMULIA 2018 , SIMULIA 2019
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	
Course Code	SIM-en-OFFSH-A-V30R2019
Available Releases	SIMULIA 2018 , SIMULIA 2019
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 indepth, 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	
Course Code	SIM-en-ADAP-A-V30R2019
Available Releases	SIMULIA 2018 , SIMULIA 2019
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

Course Code	SIM-en-SCRPT-A-V30R2019
Available Releases	SIMULIA 2018 , SIMULIA 2019
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

Available Online	Yes

Analysis of Composite Materials with Abaqus

Course Code	SIM-en-MAT-A-30-2019
Available Releases	SIMULIA 2018 , SIMULIA 2019
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 fibermatrix 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

Course Code	SIM-en-GEOT-A-V30R2019
Available Releases	SIMULIA 2018 , SIMULIA 2019
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

Prerequisites	None
Available Online	Yes

Automotive NVH with Abaqus

Course Code	SIM-en-NVH-A-V30R2019
Available Releases	SIMULIA 2018 , SIMULIA 2019
Duration	24 hours
Course Material	English
Level	Advanced
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 plugins)
Prerequisites	This course is recommended for engineers with experience using Abaqus
Available Online	Yes

Learning Experience | Course Catalog

Buckling, Postbuckling and Collapse Analysis	
Course Code	SIM-en-BUCK-A-30-2019
Available Releases	SIMULIA 2018 , SIMULIA 2019
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

Course Code	SIM-en-CMA-A-30-2019
Available Releases	SIMULIA 2018 , SIMULIA 2019
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

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

Connector Elements and Mechanism Analysis with Abaqus

Course Code	SIM-en-FLEX-A-V30R2019
Available Releases	SIMULIA 2018 , SIMULIA 2019
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

Crashworthiness Analysis with Abaqus

Course Code	SIM-en-CRASH-A-V30R2019
Available Releases	SIMULIA 2018 , SIMULIA 2019
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

Learning Experience | Course Catalog

CZone for Abaqus	
Course Code	SIM-en-CZA-A-30-2019
Available Releases	SIMULIA 2018 , SIMULIA 2019
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

Course Code	SIM-en-EMAG-A-30-2019
Available Releases	SIMULIA 2018 , SIMULIA 2019
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

Course Code	SIM-en-ELEMC-A-30-2019
Available Releases	SIMULIA 2018 , SIMULIA 2019
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

Fitness-for-Service Analysis with Abaqus

Course Code	SIM-en-FFSA-A-30-2019
Available Releases	SIMULIA 2018 , SIMULIA 2019
Duration	16 hours
Course Material	English
Level	Advanced
Audience	Fixed–Equipment Reliability Engineers
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 to assess equipment with complex geometries and loading conditions. The course discusses methods for modeling common pressure vessels such as distillation towers, storage vessels, etc. using Abaqus/CAE. Methods for the 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 discussed.
Objectives	Upon completion of this course you will be able to: - Use Abaqus/CAE to create finite element models of common plant structures. - Use Abaqus/CAE to submit and monitor analysis jobs. - Use Abaqus to perform buckling, elastic-plastic analyses.
Prerequisites	None

Fitness-for-Service Analysis with Abaqus

Available Online	Yes

GUI Customization with Abaqus

Course Code	SIM-en-GUIC-A-V30R2019
Available Releases	SIMULIA 2018 , SIMULIA 2019
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

Course Code	SIM-en-HEAT-A-V30R2019
Available Releases	SIMULIA 2018 , SIMULIA 2019
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

Course Code	SIM-en-ABI-F-30-2019
Available Releases	SIMULIA 2018 , SIMULIA 2019
Duration	32 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

Learning Experience |

Introduction to Abaqus/CAE	
Course Code	SIM-en-ICAE-F-30-2018
Available Releases	SIMULIA 2018 , SIMULIA 2019
Duration	16 hours
Course Materials	English , Japanese
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/Standard and Abaqus/Explicit

Course Code	SIM-en-IABA-F-V30R2019
Available Releases	SIMULIA 2018 , SIMULIA 2019
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

Course Code	SIM-en-ISRPT-F-V30R2019
Available Releases	SIMULIA 2018 , SIMULIA 2019
Duration	16 hours
Course Material	English
Level	Fundamental
Audience	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

Course Code	SIM-en-LNDYN-A-30-2019
Available Releases	SIMULIA 2018 , SIMULIA 2019
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	
Course Code	SIM-en-METF-A-V30R2019
Available Releases	SIMULIA 2018 , SIMULIA 2019
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

Course Code	SIM-en-METAL-A-V30R2019
Available Releases	SIMULIA 2018 , SIMULIA 2019
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 and Resolving Convergence Issues with Abaqus

Course Code	SIM-en-MCRC-A-V30R2019
Available Releases	SIMULIA 2018 , SIMULIA 2019
Duration	24 hours
Course Material	English
Level	Advanced
Audience	Simulation Analysts
Description	This 3-day course provides an in-depth discussion on solving nonlinear problems in Abaqus/Standard with an emphasis on modeling and convergence-related issues for contact. Engineers at Abaqus have developed many techniques and guidelines for solving challenging contact problems. Convergence issues related to complicated material models and geometrically unstable behavior are also covered. 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: - 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 - Understand how nonlinear problems are solved in Abaqus - Develop Abaqus models that will converge - Identify modeling errors that cause models to experience convergence difficulties

Modeling Contact and Resolving Convergence Issues with Abaqus	**32 / 87**
	- Recognize when a problem is too difficult or too illposed to be solved effectively
Prerequisites	This course is recommended for engineers with experience using Abaqus
Available Online	Yes

Modeling Contact with Abaqus/Standard

Course Code	SIM-en-CONT-A-V30R2019
Available Releases	SIMULIA 2018 , SIMULIA 2019
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

Course Code	SIM-en-FLOW-A-V30R2019
Available Releases	SIMULIA 2018 , SIMULIA 2019
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

Prerequisites	This course is recommended for engineers with experience using Abaqus
Available Online	Yes

Modeling Fracture and Failure with Abaqus	
Course Code	SIM-en-FRAC-A-30-2019
Available Releases	SIMULIA 2018 , SIMULIA 2019
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

Course Code	SIM-en-MRUB-A-V30R2019
Available Releases	SIMULIA 2018 , SIMULIA 2019
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

Course Code	SIM-en-STENT-A-V30R2019
Available Releases	SIMULIA 2018 , SIMULIA 2019
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

Obtaining a Converged Solution with Abaqus

Course Code	SIM-en-CONV-A-V30R2019
Available Releases	SIMULIA 2018 , SIMULIA 2019
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 illposed to be solved effectively
Prerequisites	This course is recommended for engineers with experience using Abaqus
Available Online	Yes

Structural-Acoustic Analysis Using Abaqus

Course Code	SIM-en-ACOU-A-30-2019
Available Releases	SIMULIA 2018 , SIMULIA 2019
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 perform: - 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

Course Code	SIM-en-SUPSUB-A-V30R2019
Available Releases	SIMULIA 2018 , SIMULIA 2019
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

Course Code	SIM-en-TIRE2-A-V30R2019
Available Releases	SIMULIA 2018 , SIMULIA 2019
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

Course Code	SIM-en-TIRE-F-V30R2019
Available Releases	SIMULIA 2018 , SIMULIA 2019
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

Course Code	SIM-en-SUBR-A-V30R2019
Available Releases	SIMULIA 2018 , SIMULIA 2019
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 userdefined elements
Prerequisites	A working knowledge of the finite element method and programming in either Fortran or C

Writing User Subroutines with Abaqus

Available Online	Yes

SIMULIA
CST Studio Site

CST Studio Suite - EMC/EMI

Course Code	SIM-en-CSTEMC-A-V30R2019
Available Release	SIMULIA 2019
Duration	12 hours
Course Material	English
Level	Advanced
Audience	This course is intended for engineers involved in electromagnetic compatibility (EMC), electromagnetic interference (EMI), electrostatic discharge (ESD), electromagnetic environmental effects (E3) and antenna integration.
Description	This course covers the EMC Module providing an overview on how simulation can be applied towards electromagnetic compatibility (EMC) design and analysis for emissions and immunity, conducted and radiated.
Objectives	The EMC Module expands upon the CORE Module basics to provide an overview on how simulation can be applied towards electromagnetic compatibility (EMC) design and analysis. Upon completion, you will understand the different solvers, sources and outputs for EMC simulation, and you will be able to set up simulations for emissions and immunity, shielding effectiveness, 3D EM and circuit co-simulations, and cable modeling.
Prerequisites	Introduction to CST Studio Suite
Available Online	Yes

CST Studio Suite - Microwave and Antenna

Course Code	SIM-en-CSTMWA-A-V30R2019
Available Release	SIMULIA 2019
Duration	7 hours
Course Material	English
Level	Advanced
Audience	Electromagnetic Simulation Analysts & Antenna Designers
Description	Throughout this course you will become familiar with the high frequency solvers used to simulate different type of antennas. This course will also discuss more specific topics like antenna matching network and SAR calculations in the human body.
Objectives	Upon completion of this course you will be able to: - - Construct & simulate antennas using different solvers. - - Visualize & extract primary results (S-Parameters, Nearfield distribution, Farfield) & more advanced results using postprocessing (i.e. Potential Bandwidth, SAR). - - Handle imported CAD models. - - Design simple matching network. - - Use the Hybrid Solver to simulate more complex RF Systems.
Prerequisites	Introduction to CST Studio Suite
Available Online	Yes

CST Studio Suite - Multiphysics

Course Code	SIM-en-CSTMPS-A-V30R2019
Available Release	SIMULIA 2019
Duration	8 hours
Course Material	English
Level	Advanced
Audience	Recommended for Electromagnetic Simulation Analysts who also need thermal and mechanical analysis as part of their workflow
Description	This course describes how to use CST's Multiphysics Studio to perform thermal and mechanical analysis. The course reviews the basic concepts of heat transfer and mechanical behaviors of materials, then explains the sources, boundary conditions, meshing and solver options available in MPS, and illustrates the steps of an EM-Thermal-Mechanical coupled simulation.
Objectives	Upon completion of the course you will: - - Obtain basic thermal and mechanical theories - - Get an overview of CST's Multiphysics Studio's capabilities - - Understand basic sources, boundary conditions, meshing options and solver options in CST Multiphysics Studio - - Be able to run an EM-Thermal-Mechanical coupled analysis
Prerequisites	Introduction to CST Studio Suite
Available Online	Yes

Introduction to CST Studio Suite

Course Code	SIM-en-CSTCOR-F-30-2018
Available Releases	SIMULIA 2018 , SIMULIA 2019
Duration	8 hours
Course Material	English
Level	Fundamental
Audience	Electromagnetic Simulation Analysts
Description	Throughout this course you will become familiar with the basic usage of CST STUDIO SUITE in terms of its modeling and simulation capabilities. Participants will learn about the general setup, submission, and result analysis procedures for high-frequency 3D electromagnetic simulations. This class will also introduce the various high frequency solvers available CST MICROWAVE STUDIO and provide a behind-the-scenes look into the FIT and FEM algorithms for the general purpose Time-Doman and Frequency-Domain solvers respectively.
Objectives	Upon copmletion of this course you will be able to:

Objectives (continued):
- Navigate the general layout of the CST SUTDIO SUITE interface
- Generate CAD geometries within the native modeling interface
- Set up the project environment with the desired units, frequency settings, background materials and boundary conditions
- Understand the various material types that exist and how to define them
- Setup excitations using lumped elements and waveguide ports
- Setup result monitors to obtain 2D/3D field data
- Choose and set up the most appropriate solver and algorithm for high frequency applications.

Introduction to CST Studio Suite	**51 / 87**
- Run Time-Domain (FIT) and Frequency-Domain (FEM) simulations including parametric sweeps and optimizations. - Analyze simulation results such as S-parameters, voltages, currents, 3D nearfields and farfields. - Extract data from the standard result set through the use of post-processing templates	
Prerequisites	None
Available Online	Yes

SIMULIA

fe-safe

Automating Analysis in fe-safe

Course Code	SIM-en-AAFS-A-V30R2019
Available Releases	SIMULIA 2018 , SIMULIA 2019
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 - Export a project used with fe-safe to replicate analysis - Use pre-scanning and group commands in a macro file - Change project settings and run analysis in a macro - Change project settings and run analysis in a batch file - Troubleshoot and customize batch files and macros
Prerequisites	Introduction to fe-safe
Available Online	Yes

Introduction to fe-safe

Course Code	SIM-en-IFES-F-V30R2019
Available Releases	SIMULIA 2018 , SIMULIA 2019
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 fesafe - 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

Introduction to fe-safe/Rubber	
Course Code	SIM-en-IFSR-A-30-2018
Available Releases	SIMULIA 2018 , SIMULIA 2019
Duration	16 hours
Course Material	English
Level	Advanced
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	56 / 87
	- Industry examples
Prerequisites	- Introduction to Abaqus - Introduction to fe-safe
Available Online	Yes

SIMULIA

Isight

Introduction to Isight

Course Code	SIM-en-ISGT-F-V30R2019
Available Releases	SIMULIA 2018 , SIMULIA 2019
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:

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	59 / 87
Prerequisites	None
Available Online	Yes

Isight Component Development

Course Code	SIM-en-ISCD-A-V30R2019
Available Releases	SIMULIA 2018 , SIMULIA 2019
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

Optimizing Engineering Methods with Isight

Course Code	SIM-en-ISOM-A-V30R2019
Available Releases	SIMULIA 2018 , SIMULIA 2019
Duration	16 hours
Course Material	English
Level	Advanced
Audience	Simulation Analysts or Scientists
Description	This course will provide a brief overview of Isight and optimization before moving on to discuss nonlinear optimization theories and applications. Topics such as techniques for design space searching, multi-objective optimization, optimization strategy, and multidisciplinary optimization will be covered during the seminar. Attendees will learn key differences between the optimization algorithms offered by Isight, how to choose the preferred method depending 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	Gain hands-on experience to understand nonlinear optimization theories and techniques: - - How does exploring the design space can assist with optimization - - The capabilities of different optimization techniques and exploration strategies - - Methodologies for Multidisciplinary Design Optimization (MDO)
Prerequisites	Introduction to Isight

Optimizing Engineering Methods with Isight

Available Online	Yes

SIMULIA
Simpack

Introduction to Simpack

Course Code	SIM-en-SMPKBA-F-V30R2019
Available Releases	SIMULIA 2018 , SIMULIA 2019
Duration	24 hours
Course Material	English
Level	Fundamental
Audience	- Simulation analysts and design engineers - Multibody simulation experts with no previous experience of Simpack
Description	This course explains the basics of multibody model set-up, simulation and postprocessing with SIMULIA Simpack.
Objectives	Upon completion of this course you will be able to: - Build multibody models in Simpack - Start the most important Simpack solvers - Postprocess the results - Parameterize models in an efficient way - Set up modular models using Simpack's substructuring concept
Prerequisites	Some familiarity with fundamental multibody concepts
Available Online	Yes

Simpack Automotive

Course Code	SIM-en-SMPKAU-A-30-2019
Available Releases	SIMULIA 2018 , SIMULIA 2019
Duration	8 hours
Course Material	English
Level	Advanced
Audience	- People who are familiar with Simpack and work in the - automotive sector
Description	This course explains how to set-up road vehicle models and introduces Simpack's Automotive elements (tires, roads, steering controllers, etc) and the Automotive Demo Wizard Database.
Objectives	Upon completion of this course you will be able to: - Build up models with automotive elements - Build up models from automotive database
Prerequisites	Introduction to Simpack
Available Online	Yes

Simpack Contact Mechanics

Course Code	SIM-en-SMPKCM-A-30-2019
Available Releases	SIMULIA 2018 , SIMULIA 2019
Duration	8 hours
Course Material	English
Level	Advanced
Audience	Multibody simulation engineers involved in contact mechanics simulation
Description	This course explains the usage of Simpack's modeling elements for contact mechanics and features that optimize the Simpack solver for generic contact simulations.
Objectives	Upon completion of this course you will be able to: - Use Simpack's modeling elements for contact mechanics and features - Optimize the Simpack solver for generic contact simulations - Have an overview on contact models and methods in Simpack - Model single-point contact search with Moved Markers - Model generic 2D and 3D Multi-point contact methods - Model contact force calculation on rigid and flexible bodies - Perform a minimum distance measurement between arbitrary surfaces - Model examples of rigid and flexible body contact
Prerequisites	Before undertaking this course, you should have completed the Introduction to Simpack training course.

Simpack Contact Mechanics

Available Online	Yes

Simpack Drivetrain

Course Code	SIM-en-SMPKDT-A-30-2019
Available Releases	SIMULIA 2018 , SIMULIA 2019
Duration	8 hours
Course Material	English
Level	Advanced
Audience	Simpack users planning to build up and analyze drivetrain models in Simpack
Description	The drivetrain training gives an overview of Simpack modeling elements and analysis methods available for drivetrain applications.
Objectives	Upon completion of this course you will be able to have an overview of drivetrain applications including: - Joints and Constraints - Bushing and bearing elements - Gear pair Force Elements with different levels of detail - Spline coupling - Drivetrain modeling guidelines and recommendations - Drivetrain modeling guidelines Linear resonance analysis - Order analysis - Operating deflection shapes
Prerequisites	Introduction to Simpack Training course successfully finished
Available Online	Yes

Simpack Flexible Bodies

Course Code	SIM-en-SMPKFB-A-30-2019
Available Releases	SIMULIA 2018 , SIMULIA 2019
Duration	8 hours
Course Material	English
Level	Advanced
Audience	Simulation Analysts
Description	This course is about structural flexibility in Simpack. It explains how to import linear finite element (FE) models into Simpack and how to model linear and nonlinear beam structures directly in Simpack.
Objectives	Upon completion of this course you will be able to: - Prepare Abaqus models for use inside Simpack (as an Abaqus substructure) - Integrate finite element models into the Simpack environment - Define appropriate interface connections to the Simpack model - Define appropriate mode selection inside the Simpack environment - Perform Simpack analysis - Perform postprocessing of flexible body results
Prerequisites	This course is recommended for engineers with experience using FEA tools and Simpack
Available Online	Yes

Simpack FlexTrack

Course Code	SIM-en-SMPKFT-A-30-2019
Available Releases	SIMULIA 2018 , SIMULIA 2019
Duration	8 hours
Course Material	English
Level	Advanced
Audience	Simpack Rail users interested in advanced train/track interaction, and engineers involved in modelling the dynamic behavior of tracks, bridges, switches and crossings.
Description	This course explains how to set up flexible rail track models in Simpack.
Objectives	Upon completion of this course you will be able to: Set up of a finite element based flexible rail track in Simpack Preparing the FE models for the import Craigh-Bampton reduction - Import of the reduced FE-Models into Simpack Selection of the appropriate FE modeling method Modeling of nonlinear railpads and ballast defects Take advantage of Simpack scripts for semi-automatic flexible track setup
Prerequisites	Introduction to Simpack Course and the Simpack Rail course; the Simpack Flexible Bodies course is beneficial but not mandatory.
Available Online	Yes

Simpack NVH

Course Code	SIM-en-SMPKNV-A-30-2019
Available Release	SIMULIA 2019
Duration	8 hours
Course Material	English
Level	Advanced
Audience	
Description	This course covers the fundamental theory necessary to understand NVH, along with a detailed look at the NVH functionality within Simpack. Using prepared models, participants have the opportunity to experiment with and compare the various methods.
Objectives	Upon completion of this course you will be able to: - Understand NVH fundamentals and application areas: - Understand NVH functionality within Simpack - Frequency response functions - Linear system response - Spectral analysis - Frequency sweep - Nonlinear Frequency Pass - Order Analysis - Operating Deflection Shapes - Use these methods with complete road and rail vehicles
Prerequisites	
Available Online	Yes

Simpack Rail	
Course Code	SIM-en-SMPKRA-A-30-2019
Available Releases	SIMULIA 2018 , SIMULIA 2019
Duration	16 hours
Course Material	English
Level	Advanced
Audience	- Simulation analysts and design engineers in the railway industry - Multibody simulation experts with no previous experience of railway simulation in Simpack
Description	The Simpack Rail training explains how to use Modeling Elements to simulate rail-wheel contact, how to set up entire vehicle models, and the most important analysis types for rail vehicles. It also introduces rail vehicle specific elements such as tracks, irregularities, wheel and rail profiles, suspension elements.
Objectives	Upon completion of this course you will be able to: - Understand all necessary elements for railway modeling - Set up railway vehicles according to common concepts - Set up, run and analyze typical applications for railway models
Prerequisites	- Before undertaking this course, you should have completed the Introduction to Simpack training course - Some familiarity with fundamental railway theory
Available Online	Yes

Simpack Scripting

Course Code	SIM-en-SMPKSC-A-30-2019
Available Releases	SIMULIA 2018 , SIMULIA 2019
Duration	8 hours
Course Material	English
Level	Advanced
Audience	Simpack users wishing to automate any tasks in Simpack
Description	This course explains how to automate tasks in Simpack Pre and Post through scripting. It provides user experience in terms of writing, running and debugging scripts.
Objectives	This course provides an overview of the Simpack scripting capabilities and possible applications. Upon completion of this course you will be able to: - Understand the basics of the scripting languages. - Use Simpack-specific scripting classes. - Create examples for the most common scripting applications. - Write, run and debug scripts.
Prerequisites	Completion of the Simpack Basics training course. Basic programming knowledge is required.
Available Online	Yes

Simpack User Routines

Course Code	SIM-en-SMPKUR-A-30-2019
Available Releases	SIMULIA 2018 , SIMULIA 2019
Duration	8 hours
Course Material	English
Level	Advanced
Audience	Simpack users wishing to extend Simpack's built-in functionality by user written Modeling Elements will benefit from this course.
Description	The User Routines training explains how to add userdefined functionality to Simpack by means of Fortran subroutines.
Objectives	Upon completion of this course you will be able to: - Define element parameters including selection menus and units - Program a force law using Simpack states - Program a Simpack filter - Incorporate external routines and libraries into Simpack - Debug user programmed elements
Prerequisites	Completion of the Introduction to Simpack training course. Basic programming skills are recommended
Available Online	Yes

Learning Experience | Course Catalog

Simpack Wind	
Course Code	SIM-en-SMPKWD-A-30-2019
Available Releases	SIMULIA 2018 , SIMULIA 2019
Duration	8 hours
Course Material	English
Level	Advanced
Audience	Simpack users planning to build up and analyze wind turbine models in Simpack.
Description	This course explains how to set up wind turbine models in Simpack, how to use the rotorblade generator with linear and nonlinear SIMBEAM elements, wind specific elements (including the interfaces to AeroDyn and HydroDyn), and how to use the Simpack wind load calculation tool.
Objectives	Upon completion of this course you will be able to: - Understand Simpack Wind turbine applications - Model structural components (onshore and offshore foundation, tower, nacelle) - Generate and model Rotorblades in Simpack - Conduct Aero- and Hydro-elastic simulation - Use the Load Calculations Tool - Model high fidelity drivetrains - Conduct a resonance analysis according to GL 2010
Prerequisites	Introduction to Simpack and Simpack Flexible Body Trainings.
Available Online	Yes

SIMULIA

Tosca Structure

Learning Experience | Course Catalog

Introduction to Tosca Structure	
Course Code	SIM-en-TOSCST-F-V30R2019
Available Releases	SIMULIA 2018 , SIMULIA 2019
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

SIMULIA
XFlow

Learning Experience | Course Catalog

Introduction to XFlow	
Course Code	SIM-en-XFINT-F-30-2018
Available Release	SIMULIA 2018
Duration	8 hours
Course Material	English
Level	Fundamental
Audience	Engineers with knowledge or experience in CFD
Description	This course introduces the XFlow numerical methodology, the Graphical User Interface (GUI), and shows how to get started with the software.
Objectives	Upon completion of this course you will be able to: - Understand the Lattice-Boltzmann Method used in XFlow - Start with the XFlow GUI - Set up a simulation in XFlow - Know how to handle geometries in XFlow - Use input functions - Postprocess simulations in XFlow
Prerequisites	None
Available Online	Yes

XFlow Acoustics Analysis

Course Code	SIM-en-XFACA-A-30-2018
Available Release	SIMULIA 2018
Duration	3 hours
Course Material	English
Level	Advanced
Audience	Engineers with knowledge or experience in CFD
Description	This course explains the method used in XFlow to carry out acoustics analysis.
Objectives	Upon completion of this course you will be able to: - Understand the acoustics methods used in XFlow - Carry out acoustics simulations in XFlow - Use the acoustics postprocessing features of XFlow
Prerequisites	- Before taking this course the completion of the following prequisite courses (or equivalent knowledge) is required: - XFlow Introduction - XFlow Single Phase Flow
Available Online	Yes

XFlow Advanced Capabilities	
Course Code	SIM-en-XFADV-A-30-2018
Available Release	SIMULIA 2018
Duration	8 hours
Course Material	English
Level	Advanced
Audience	Engineers with knowledge or experience in CFD
Description	This course introduces the advanced features and scripting capabilities of XFlow
Objectives	Upon completion of this course you will be able to: - Carry out supersonic simulations in XFlow - Use Porous Media and Porous Jump boundary conditions in XFlow - Use the XFlow Scalar Transport model - Use the XFlow Discrete Phase Model (DPM) - Apply Turbulence Intensity laws in external simulations - Execute XFlow simulations using command lines - Use scripts to automate the execution and postprocessing of XFlow simulations - Run serial or parallel XFlow simulations on HPC
Prerequisites	- Before taking this course the completion of the following prequisite courses (or equivalent knowledge) is required: - XFlow Introduction - XFlow Single Phase Flow - XFlow Free-surface Flow
Available Online	Yes

Learning Experience | Course Catalog

XFlow Complete	
Course Code	SIM-en-XFLOW-A-30-2018
Available Release	SIMULIA 2018
Duration	40 hours
Course Material	English
Level	Advanced
Audience	Engineers with knowledge or experience in CFD
Description	This course covers all the XFlow capabilities and how to set up, and postprocess all kind of simulations in XFlow
Objectives	Upon completion of this course you will be able to: - Understand the Lattice-Boltzmann Method used in XFlow - Use the different features available for all XFlow solvers: Single Phase, Free- surface and Multiphase solvers - Set up internal and external simulations using all the solvers of XFlow - Set up thermal and acoustics analysis in XFlow - Set up simulations in XFlow with different moving parts behaviors - Handle all the postprocessing tolls of XFlow - Use advanced features and scripting capabilities of XFlow
Prerequisites	None
Available Online	Yes

XFlow Free-surface Flow

Course Code	SIM-en-XFFSF-A-30-2018
Available Release	SIMULIA 2018
Duration	3 hours
Course Material	English
Level	Advanced
Audience	Engineers with knowledge or experience in CFD
Description	This course introduces the Xflow Free-surface solver features and covers how to set up simulations with this solver
Objectives	Upon completion of this course you will be able to: - Understand different features of XFlow Free-surface solver - Understand different applications of XFlow Free-surface solver - Set up an external and internal Free-surface simulation in XFlow - Postprocess an external and internal Free-surface simulation in XFlow
Prerequisites	- Before taking this course the completion of the following prerequisite courses (or equivalent knowledge) is required: - XFlow Introduction
Available Online	Yes

XFlow Moving Parts

Course Code	SIM-en-XFMPA-A-30-2018
Available Release	SIMULIA 2018
Duration	8 hours
Course Material	English
Level	Advanced
Audience	Engineers with knowledge or experience in CFD
Description	This course covers how to handle and set up simulations in XFlow with different moving parts behaviors
Objectives	Upon completion of this course you will be able to: - Understand the difference between all XFlow geometries behaviors - Set up simulations in XFlow with the Enforced geometry behavior - Set up simulations in XFlow with the Rigid Body Dynamics geometry behavior
Prerequisites	- Before taking this course the completion of the following prequisite courses (or equivalent knowledge) is required: - XFlow Introduction
Available Online	Yes

XFlow Multiphase Flow

Course Code	SIM-en-XFMPF-A-30-2018
Available Release	SIMULIA 2018
Duration	5 hours
Course Material	English
Level	Advanced
Audience	Engineers with knowledge or experience in CFD
Description	This course introduces the Xflow Multiphase solver features and covers how to set up simulations with this solver
Objectives	Upon completion of this course you will be able to: - Understand the differences between the 3 Multiphase solvers available in XFlow: Particle-Based Tracking, Phase Field and Volume of Fluid - Understand the features of all XFlow Multiphase solvers - Use different applications of XFlow Multiphase solvers - Set up an external and internal Multiphase simulations in XFlow - Postprocess an external and internal Multiphase simulation in XFlow
Prerequisites	- Before taking this course the completion of the following prequisite courses (or equivalent knowledge) is required: - XFlow Introduction
Available Online	Yes

XFlow Single Phase Flow

Course Code	SIM-en-XFSPF-A-30-2018
Available Release	SIMULIA 2018
Duration	3 hours
Course Material	English
Level	Advanced
Audience	Engineers with knowledge or experience in CFD
Description	This course introduces the features of the Single Phase solver in XFlow and covers how to set up simulations using this solver
Objectives	Upon completion of this course you will be able to: - Understand different features of XFlow Single Phase solver - Use different applications of XFlow Single Phase solver - Set up an external and internal subsonic single phase simulation in XFlow - Postprocess an external and internal single phase simulation in XFlow
Prerequisites	- Before taking this course the completion of the following prequisite courses (or equivalent knowledge) is required: - XFlow Introduction
Available Online	Yes

XFlow Thermal Analysis

Course Code	SIM-en-XFTHA-A-30-2018
Available Release	SIMULIA 2018
Duration	3 hours
Course Material	English
Level	Advanced
Audience	Engineers with knowledge or experience in CFD
Description	This course explains the differences between the thermal models available in XFlow and how to use them
Objectives	Upon completion of this course you will be able to: - Understand the difference between each thermal model available in XFlow - Use each thermal model in XFlow - Carry out Conjugate Heat Transfer analysis - Use Radiation model in XFlow - Postprocess thermal simulations in XFlow
Prerequisites	- Before taking this course the completion of the following prequisite courses (or equivalent knowledge) is required: - XFlow Introduction - XFlow Single Phase Flow
Available Online	Yes