Fluid Dynamics Engineer Essentials

R2020x
Course objectives
Upon completion of this course you will be able to:

- Set up and create CFD simulations in the 3DEXPERIENCE platform
- Perform incompressible and compressible CFD analyses
- Perform fully coupled conjugate heat transfer (CHT) analyses
- Postprocess results

Targeted audience
This course is intended for the following roles:

- Fluid Dynamics Engineer
- Multiscale Systems Specialist

Prerequisites
The following course is required prior to taking this one:

- None
Day 1

- Lesson 1: Introduction to the 3DEXPERIENCE Platform
- Workshop 1a: Getting Started with the 3DEXPERIENCE Platform
- Workshop 1b: Fluid Flow in the rear duct of an automotive HVAC system
- Lesson 2: Fluid Dynamics Engineer Role Overview
- Lesson 3: Getting Started with CFD Simulation
- Workshop 2: CFD Analysis of an Air Intake System
- Lesson 4: Geometry for CFD Simulations
- Lesson 5: Meshing for CFD Simulations
- Workshop 3: Steady-state External Flow over a Drone in Cruise
- Lesson 6: Material and Section Properties
- Lesson 7: Defining Physics
- Workshop 4: Conjugate Heat Transfer (CHT) Analysis of an Electronics Module
Day 2

- Lesson 8  Boundary and Initial Conditions
- Lesson 9  Turbulence Modeling
- Workshop 5  Unsteady Flow across a Circular Cylinder
- Lesson 10  Modeling Techniques
- Workshop 6  Transonic Flow over an Airfoil
- Lesson 11  Solution Convergence
- Lesson 12  Running Simulations
- Workshop 7  Cavitating Flow through a Narrowing Pipe
- Lesson 13  Postprocessing Results
Additional Material

- Appendix 1: Review of CFD Fundamentals
- Appendix 2: Geometric Operations
- Workshop 8: Defeaturing of a Lens Component
Join the Community!

How can you maximize the robust technology of the SIMULIA Portfolio?
Connect with peers to share knowledge and get technical insights

Go to [www.3ds.com/slc](http://www.3ds.com/slc) to log in or join!

---

Let the SIMULIA Learning Community be Your Portal to 21st Century Innovation

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

For more information and registration, visit [3ds.com/simulia-learning](http://3ds.com/simulia-learning).


---

©2013 Dassault Systèmes. All rights reserved.
SIMULIA Training

http://www.3ds.com/products-services/simulia/services/training-courses/

SIMULIA SERVICES
PROVIDING HIGH QUALITY SIMULATION AND TRAINING SERVICES TO ENABLE OUR CUSTOMERS TO BE MORE PRODUCTIVE AND COMPETITIVE.

Training Schedule & Registration
We offer regularly scheduled public seminars as well as training courses at customer sites. An extensive range of courses are available, ranging from basic introductions to advanced courses that cover specific analysis topics and applications. On-site courses can be customized to focus on topics of particular interest to the customer, based on the customer’s prior specification. To view the worldwide course schedule and to register for a course, visit the links below.

North American
- By Location
- By Course

International
- By Location
- By Course

Live Online Training
- Full Schedule
The software described in this documentation is available only under license from Dassault Systèmes or its subsidiaries and may be used or reproduced only in accordance with the terms of such license.

This documentation and the software described in this documentation are subject to change without prior notice.

Dassault Systèmes and its subsidiaries shall not be responsible for the consequences of any errors or omissions that may appear in this documentation.

No part of this documentation may be reproduced or distributed in any form without prior written permission of Dassault Systèmes or its subsidiaries.

© Dassault Systèmes, 2020

Printed in the United States of America.

Abaqus, the 3DS logo, and SIMULIA are trademarks or registered trademarks of Dassault Systèmes or its subsidiaries in the US and/or other countries.

Other company, product, and service names may be trademarks or service marks of their respective owners. For additional information concerning trademarks, copyrights, and licenses, see the Legal Notices in the 3DEXPERIENCE User Assistance.
# Revision Status

| Lesson 1  | 4/20 | Updated for R2020x |
| Lesson 2  | 4/20 | Updated for R2020x |
| Lesson 3  | 4/20 | Updated for R2020x |
| Lesson 4  | 4/20 | Updated for R2020x |
| Lesson 5  | 4/20 | Updated for R2020x |
| Lesson 6  | 4/20 | Updated for R2020x |
| Lesson 7  | 4/20 | Updated for R2020x |
| Lesson 8  | 4/20 | Updated for R2020x |
| Lesson 9  | 4/20 | Updated for R2020x |
| Lesson 10 | 4/20 | Updated for R2020x |
| Lesson 11 | 4/20 | Updated for R2020x |
| Lesson 12 | 4/20 | Updated for R2020x |
| Lesson 13 | 4/20 | Updated for R2020x |
| Appendix 1| 4/20 | Updated for R2020x |
| Appendix 2| 4/20 | New for R2019x    |

| Workshop 1a | 4/20 | Updated for R2020x |
| Workshop 1b | 4/20 | Updated for R2020x |
| Workshop 2  | 4/20 | Updated for R2020x |
| Workshop 3  | 4/20 | Updated for R2020x |
| Workshop 4  | 4/20 | Updated for R2020x |
| Workshop 5  | 4/20 | Updated for R2020x |
| Workshop 6  | 4/20 | Updated for R2020x |
| Workshop 7  | 4/20 | Updated for R2020x |
| Workshop 8  | 4/20 | New for R2019x    |
Lesson 1: Introduction to the 3DEXPERIENCE Platform

Lesson content:

- What is the 3DEXPERIENCE Platform?
- Architecture
- Packaging
- 3DEXPERIENCE Platform Services
- Deployment Options
- Connecting to the Platform
- Platform Interface
- Importing and Exporting Data
- Searching Data
- Exploring Data
- Managing Data
- Compute Orchestration Services
- Simulation Conventions in the 3DEXPERIENCE Platform
- Workshop Preliminaries

1 hour
In this workshop, you will use an assembly of an automotive HVAC (Heating, Ventilation and Air-Conditioning) system to get familiar with the 3DEXPERIENCE platform and learn basic operations. This workshop can be used as a segue to Workshop 1b.

After completion of this exercise, you will be able to:

a. Navigate through the 3DEXPERIENCE platform interface
b. Import an assembly of parts
c. Explore an assembly of parts
d. Rename objects
e. Change the look of the authoring window
f. Search an assembly of parts in the database
g. Duplicate, delete and save entities
h. Import a 3D XML file containing fluid materials
i. Search a material in the database
j. Create and save a new material
In this workshop, you will simulate three-dimensional turbulent fluid flow in the rear duct of an automotive HVAC system. Later, you will also modify the outlet cross section of the duct and assess its impact on the duct performance. Such workflows are commonly found when designing ducts for an automotive HVAC system. A steady-state incompressible fluid flow will be performed.

After completion of this exercise, you will be able to:

a. Search and open an assembly in the 3DEXPERIENCE platform
b. Create a fluid domain for an internal flow simulation
c. Set up a steady-state incompressible fluid flow analysis
d. Apply boundary conditions required for CFD simulation
e. Mesh the fluid domain with a hex-dominant boundary layer mesh
f. Postprocess the results
g. Modify the outlet cross-section via parameters and re-run the simulation
h. Compare differences across both the designs and draw conclusions
Lesson 2: Fluid Dynamics Engineer Role Overview

Lesson content:

- Fluid Dynamics Engineer Role
- CFD Simulation Workflow
- Model Preparation
- Materials
- Meshing
- CFD Analysis
- Co-simulation Analysis
- Multiphysics Multiscale
- Results Postprocessing
- CFD Solver Validation

1 hour
Lesson content:

- Creating a CFD Simulation
- Fluid Scenario Creation App Interface
- Finite Element Model Representation
- Model Setup
- Scenario Setup
- Results Visualization
- Reviewing Simulation Features

Lesson 3: Getting Started with CFD Simulation

1.5 hours
In this workshop, you will simulate three-dimensional turbulent fluid flow in an air intake assembly. The air intake assembly consists of an air filter to clean the incoming air. A steady-state incompressible fluid flow analysis in conjunction with porous media modeling will be performed.

After completion of this exercise, you will be able to:

a. Search and open an assembly in the 3DEXPERIENCE platform
b. Create a fluid domain and a porous domain for an internal flow simulation
c. Mesh the fluid and porous domains with hex-dominant and sweep 3D meshes, respectively
d. Set up steady-state incompressible fluid flow analysis for a porous medium
e. Apply initial and flow conditions required for CFD simulation
f. Postprocess the results
Lesson content:

- Model Setup Overview
- Setting up a FEM Rep
- Setting up a Fluid Volume
- Geometry Preparation
Lesson 5: Meshing for CFD Simulations

Lesson content:

- CFD Meshing Basics
- Creating Meshes
- Managing Meshes
- Updating Meshes
- Excluding Meshes from a Simulation
- Importing and Exporting Meshes
- CFD Meshing
- Hex-dominant Mesh
- Partition Hex Mesh
- Octree Tetrahedron Mesh
- Tetrahedron Mesh
- Tetrahedron Filler Mesh
- Sweep 3D Mesh
- Surface Meshing
- Visualizing the Mesh
- Reviewing the Mesh
- Groups
- Selection Tools

90 minutes
In this workshop, you will simulate three-dimensional turbulent fluid flow over a drone model under cruise conditions. Such workflows are commonly found when performing aerodynamic performance analysis early in the concept design of drones. A steady-state incompressible fluid flow simulation will be performed.

After completion of this exercise, you will be able to:

- a. Import a model into the 3DEXPERIENCE platform
- b. Create a fluid domain for an exterior flow simulation
- c. Set up a steady-state incompressible fluid flow analysis
- d. Apply boundary conditions required for CFD simulation
- e. Mesh the fluid domain with a hex-dominant boundary layer mesh
- f. Postprocess the results and draw conclusions
Lesson 6: Material and Section Properties

Lesson content:

- Understanding Materials
- Working with Materials
- Available Materials
- Creating a New Material
- Applying a Material
- Adding New Domains
- Editing a Material Domain
- Simulation Domains
- Material Behaviors in a Simulation Domain
- Section Properties
- Working with Imported Meshes

1 hour
Lesson 7: Defining Physics

Lesson content:

- FEM Rep for Simulation
- Simulation Scenario Setup
- Defining Physics
- Analysis Procedures
- Steady-state Analysis
- Transient Analysis
- Bad Cell Treatment
In this workshop, you will simulate the air flow and heat transfer within an electronics module. The assembly contains several components such as a circuit board, chips, heat sinks, capacitors and a transformer. The thermal power of the components is specified. The module is cooled by a fan. After completion of this exercise, you will be able to:

- Create automatic fluid volumes for complex assemblies.
- Create non-conformal meshes with appropriate meshing techniques.
- Create a fully-coupled conjugate heat transfer analysis comprising of multiple solid domains and a fluid domain.
- Model steady-state turbulent flow using the $k-\varepsilon$ model.
- Create automatic solid-solid and solid-fluid thermal interfaces.
- Model the effect of a fan through a fan curve.
- Model interfacial thermal resistance between components.
- Postprocess the results.

Workshop 4: Conjugate Heat Transfer (CHT) Analysis of an Electronics Module

1 hour
Lesson content:

- Boundary Conditions
- Initial Conditions
- Initializing Compressible Flows
- Turbulence Specifications at Boundaries
- Surface-to-surface Radiation Specifications at Boundaries
- Time-dependent Boundary Conditions
- Spatially-varying Boundary Conditions
- User-defined Boundary Conditions
Lesson 9: Turbulence Modeling

Lesson content:

- What is Turbulence?
- Is the Flow Turbulent?
- Turbulence Modeling
- How Realistic does the Solution Look?
- Activating Turbulence Models
- Near-Wall Modeling
- Turbulence Initial Conditions
- Turbulence Boundary Conditions
The phenomenon of vortex shedding is important in engineering applications such as heat exchangers, nuclear reactor fuel rod assemblies, suspension bridge and other numerous applications. For flow passing over a stationary cylinder, experimental observations and numerical predictions have shown that a vortex street in the wake of the cylinder is formed which induces unsteady lift and drag forces on the cylinder. The unsteadiness in the fluid forces can induce vibrations on structures which need to be considered during their design.

In this workshop, we analyze the unsteady flow across a circular cylinder at a Reynolds number of 100. This classical problem forms the basis of many engineering problems where both the vortex generation as well as vortex-induced vibrations need to be considered.

After completion of this exercise, you will be able to:

a. Set up a flow simulation with an available fluid domain geometry.
b. Mesh the fluid domain with a sweep 3D mesh.
c. Set up a transient incompressible fluid flow simulation to study vortex shedding phenomena.
d. Apply initial and flow conditions required for CFD simulation.
e. Postprocess the results.
Lesson content:

- Contact Interfaces
- Porous Media Modeling
- Multiple Reference Frames
- Multiphase Flows
- Multispecies Flows
- Modeling Conjugate Heat Transfer
- Modeling Joule Heating
- Modeling Radiation
- Modeling Solar Radiation
- Flows with Moving/Deforming Boundary
- Modeling Generic Body Forces
- Modeling Sources
- Time-dependent Boundary Conditions
- Spatially-varying Mapped Data
- User-defined Subroutines
In this workshop, you will simulate two-dimensional turbulent transonic airflow over an airfoil (RAE 2822) at an attack angle of 2.31°. The free-stream Mach number for the flow is 0.729. At these settings, a shock wave is produced on the top of the airfoil. You will also compare the CFD results to available experimental results.

After completion of this exercise, you will be able to:

a. Import an orphan mesh to use in simulation.
b. Model compressible flow.
c. Use the Ideal Gas Law to define the properties of air.
d. Use the one-equation Spalart-Allmaras turbulence model.
e. Monitor solution convergence using force monitors.
f. Postprocess results for compressible flows.

Lesson 11: Solution Convergence

Lesson content:

- Solution Convergence
- Solver Controls
- Diagnosing Convergence Issues
- Iterative Techniques
- Algebraic Multigrid (AMG) Technology
Lesson 12: Running Simulations

Lesson content:

- Solution Output
- Running Simulations
- Viewing Diagnostics
- Managing Simulations
- Restarting Simulations
- Setting Visualization Options

1 hour
In this workshop, you will simulate the pressure-driven cavitating flow of water through a tube that narrows in its middle section. Many industrial applications have a set up like this and accurately modeling the physics of cavitation is essential to ensuring product durability. Due to the high pressure differential, cavitation can occur leading to the formation of a vapor phase which can damage critical components in the assembly. A CAD model representing the tube section is used to study the cavitating flow of water. A transient fluid flow simulation will be performed using the volume of fluids (VOF)-based multiphase modeling capability. A quarter-symmetry fluid domain of the model will be used in the simulation.

After completion of this exercise, you will be able to:

a. Create a fluid domain for internal flow simulation with a quarter-symmetry domain
b. Mesh the fluid domain with a hex-dominant boundary layer mesh
c. Create a VOF section for multiphase analysis involving two water phases, namely liquid water and water vapor
d. Set up a transient incompressible fluid flow simulation with multiphase modeling
e. Apply initial and boundary conditions required for multiphase simulations
f. Postprocess the results
Lesson 13: Postprocessing Results

Lesson content:

- Accessing Results
- Contour Plots
- Symbol Plots
- Isocontour Plots
- Mirroring and Patterning Results
- Activating Multiple Plots
- Showing Min/Max Values
- Controlling Plot Legends
- Plot Sections
- Stream Toolset (Instantaneous Particle Traces)
- Path Plots
- Field Expressions
- Rendering Settings
- Display Groups
- Animations
- Creating Reports

1 hour
Appendix 1: Review of CFD Fundamentals

Appendix content:

- Overview
- What is CFD?
- Numerical Simulation of Physical Phenomena
- Computational Solid Mechanics (CSM) vs. Computational Fluid Dynamics (CFD)
- CFD Basics
- Governing Equations
- Diffusion and Advection
- Flow Features
- Heat Transfer in Fluid Dynamics
- Non-dimensional Quantities in CFD
- Initial and Flow Conditions
- Solution Methodology
- Turbulence Modeling
- References

1.5 hours
Appendix 2: Geometric Operations

Appendix content:

- Introduction
- Accessing the Simulation Model Preparation app
- The Simulation Model Preparation app
- Geometric Checkers
- Join
- Healing
- Remove Face
- Replace Face
- Defeaturing
- Split
- Disconnect
- Partition
- Extract
- Trim
- Mid Surface
- Close Surface
- Projection
- Visualization Management
- Clipping Box
- Meshability
- Status Bar

1.5 hours
Workshop 8: Defeaturing of a Lens Component

In this workshop, you will import a part into the 3DEXPERIENCE platform. The CAD data represent a lens component provided in STP format. After importing the part, you will switch to the Simulation Model Preparation app and defeature the geometry.

After completing this exercise, you will be able to:

a. Import a neutral format CAD file.

b. Use the Simulation Model Preparation app to defeature the geometry.