

TRAINING COURSE

OCTOBER 15–18 11AM–5PM EDT

CFD MODELING IN COMSOL MULTIPHYSICS® FOR CMC

SCHEDULE

Day 1

The training course begins with an exercise of putting together a simple single-phase flow model to familiarize ourselves with the user interface (UI) and the COMSOL Multiphysics® modeling workflow. The following topics will be covered:

- The COMSOL Multiphysics® UI and workflow for setting up laminar flow models
- Simple theoretical overview of governing equations
- Flow regimes
- Boundary conditions
- Non-Newtonian flow
- Turbulent flow
- General convergence techniques

Day 2

The second day builds upon the training from the previous day. Topics will include:

- Compressible flow
- Conjugate heat transfer
- Porous media flow
- Reacting flow
- FSI

Day 3

- Work on exercises (no class)

Day 4

The third day will focus on the methods available in the COMSOL® software for modeling multiphase flow. Topics will include:

- Separated multiphase flow
- Dispersed multiphase flow

SUGGESTED BACKGROUND

This course assumes familiarity with the fundamentals of computational fluid dynamics. We strongly recommend that those new to COMSOL Multiphysics® take the Introduction to COMSOL Multiphysics® training course prior to attending this class.

The CFD modeling course will provide you with a comprehensive introduction to the CFD Module, an add-on to the COMSOL Multiphysics® software that provides capabilities for modeling a wide range of flow phenomena. During the 4-day online course, we will introduce you to various physics interfaces and general functionality provided in this module. By the end of the course you will have been introduced to single-phase laminar flow modeling, turbulent flow, compressible flow, conjugate heat transfer, reacting flow, fluid–structure interaction (FSI), multiphase flow, and porous media flow. In addition, we will discuss best practices and useful convergence techniques depending on the specific type of problem at hand.

FOR QUESTIONS PLEASE CONTACT
James.Christopher@comsol.com

