During the 2-day COMSOL Multiphysics® Fluid Flow and Heat Transfer Intensive Training course, you will develop a strong foundation for your fluid flow and thermal modeling work.

We use a combination of guided hands-on training, theoretical and practical lectures, and self-guided hands-on training to teach this course. The goal is to immerse you in all of the main aspects of using COMSOL Multiphysics® and the add-on modules for fluid flow and heat transfer, so that you feel comfortable working with the software. By the end of the training, you will know how to set up a fluid flow or thermal problem, choose the right boundary conditions, select appropriate mesh settings, and use the result visualization tools to create stunning graphics.

Day 1, Fluid Flow, 09:00 - 17:00

The course begins with a guided introduction to the graphical user interface (GUI) of the software, so that you learn the key steps in the modeling workflow. This day 1 focuses on fluid flow.

- Introduction to fluid flow with COMSOL Multiphysics
- Preparing CAD imports for flow simulations
- Best practices for meshing
- Free and porous media flow
- Determining the laminar/turbulent flow regime
- Selecting the right turbulence model and working with it
- Multiphysics effects like thermo-fluid dynamics, fluid-structure interaction and Marangoni convection
- Microfluidics
- Specialized boundary conditions like fans, screens, grilles, interior walls, and pumps
- Rotating machinery
- Pipe flow
- Particle tracing in fluids
- Error messages related to convergence issues

Day 2, Heat Transfer, 09:00 - 17:00

The second day builds on day one, but with heat transfer in focus:

- Introduction to heat transfer with COMSOL Multiphysics
- Balance equations
- Modeling conduction, convection, and radiation
- Free and natural convection
- Multiphysics effects like thermo-fluid dynamics, electromagnetic heating, thermal stress and chemical reactions
- Specialized boundary conditions
- Thin layers, thermal contact, infinite elements and other special features
- Turbulent heat transfer
- Phase change

Suggested Background

Some prior knowledge of the basic concepts of fluid flow and heat transfer is recommended. To get the most out of the training, you should either have been using COMSOL Multiphysics® in the past or attended a course such as the COMSOL Multiphysics® Essential Training course.