Heat Transfer Modeling in COMSOL Multiphysics®

SciEngineer’s training courses are designed to help organizations and individuals close skills gaps, keep up-to-date with the industry-accepted best practices and achieve the greatest value from MathWorks® and COMSOL® Products.

Heat Transfer Modeling in COMSOL Multiphysics®

If you are interested in solving realistic heat transfer problems using COMSOL Multiphysics®, this is the course for you. During this 2-day course, we will cover the modeling of heat transfer via conduction, convection, and radiation in solids and fluids. Modeling of temperature fields in solids, liquids, gases, porous materials, thin-walled parts, and pipes will be addressed. Additionally, you will learn about modeling Multiphysics problems in which temperature fields are computed along with electromagnetic fields, structural deformations, and other physics. To teach this course, we complement hands-on problem solving with discussions of relevant theory and best practices. By the end of the course, you will know how to successfully and efficiently set up heat transfer simulations using the COMSOL® software.

Suggested Background

This course assumes some familiarity with the basic concepts of heat transfer. We strongly recommend that those new to COMSOL Multiphysics® take the COMSOL Multiphysics® Intensive course prior to attending this class.

Topics Include

- Steady-state, transient, and frequency domain solutions
- Modeling of liquids and gases via explicit and empirical approaches
- Free- and forced-convection modeling
- Bioheat transfer
- Electromagnetic heating
- Heat transfer in porous media
- Surface-to-ambient and surface-to-surface radiation
- Phase change processes

Expand your knowledge