

COMSOL Multiphysics® CFD & Thermal Simulations

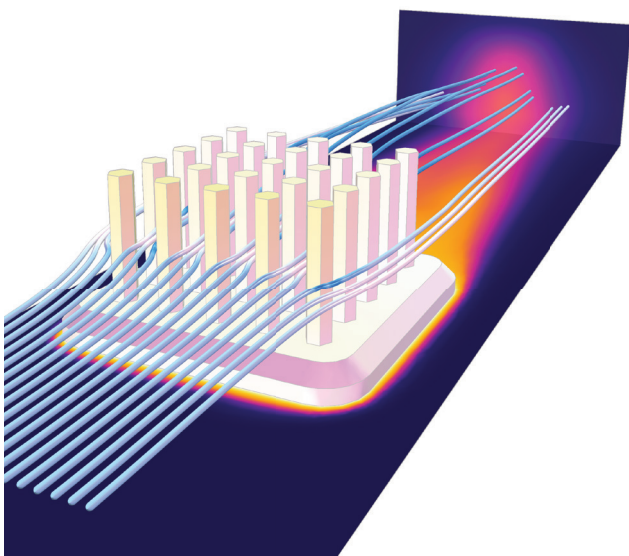
WORKSHOP

Tuesday,
March 7

1:00PM–4:00PM

LOCATION

The Yale Center for Research Computing
YCRC Auditorium
160 Ronan Street
New Haven, Connecticut



Free of charge and a two-week trial of COMSOL Multiphysics is available upon request

During this workshop you will:

- Get an interactive introduction to COMSOL Multiphysics® software.
- Discover the extensive capabilities of the software.
 - Model laminar and turbulent flow, single and multiphase flow, non-Newtonian flow, and porous media flow.
 - Predefined multiphysics such as fluid-structure interaction, non-isothermal flow, reacting flow, and particle tracing.
 - Model thermal conduction, convective heat transfer, and radiative heat transfer.
 - Analyze heat transfer in porous media, bioheating, phase change, and thermal contact resistance.
 - Couple heat transfer with any other physics, such as fluid flow, chemical reactions, structural mechanics, or electromagnetics.
- Learn how to design a custom simulation app with the Application Builder.
- Discuss your projects with a COMSOL® support engineer.

SPEAKER

Angela Straccia, COMSOL, Inc.
Mats Gustafsson, COMSOL, Inc.

DETAILS & REGISTRATION

<http://comsol.com/c/4ztd>

