

Multiphysics simulation of piezoelectric sensors and actuators

David W. Greve

**Emeritus Professor of Electrical and Computer Engineering, Carnegie Mellon
University, Pittsburgh, PA, USA**

Principal, DWGreve Consulting

This course will focus on the use of finite element simulations to predict and understand piezoelectric materials as used in sensors and actuators. Real-world applications involve complex geometries not suitable for analytic approaches and in addition involve wave propagation into liquids, gases, or other solid materials. Finite element simulation allows us to analyze problems of this type.

The finite element application COMSOL will be used in the course. Topics will include: setting up and running finite element simulations of ultrasonic transducers; post-processing of the results; when to use eigenfrequency, frequency domain, and time dependent studies; and development of multiphysics models where different physics apply in different parts of the simulation domain.

The course includes both a tutorial and a survey component. The details in setting up selected models will be presented and the reasoning behind selections of boundary conditions, mesh type and sizing, multiphysics couplings, and type of simulation study will be explained. Results from multiphysics simulations will be presented including: piezoelectric transducers coupled to a solid domain; surface acoustic wave devices and coupling of surface waves into fluids; and forces on objects in ultrasonic fields. These examples will illustrate the range of problems that can be addressed by finite element simulations.

The instructor will provide a time-limited evaluation license to registered attendees, who will be able to download and run COMSOL. Setup files for many of the simulations discussed will be made available to the participants.