

Tutorium “CFD-based simulation and optimization of microfluidic components”

In the past decades, the field of microfluidics has undergone a rapid development, with a number of emerging application areas such as lab-on-a-chip technology. In contrast to their macroscopic counterparts, flows in microfluidic devices are mostly laminar. This opens up unprecedented possibilities of designing and optimizing these devices “in silico”, since modeling and simulation based on first-principle theoretical descriptions such as the Navier-Stokes equation instead of heuristic models such as turbulence models becomes possible.

The new tutorial “CFD-based simulation and optimization of microfluidic components”, offered by the Institute of Nano- and Microfluidics, will equip students with the basic methodology needed to design and optimize microfluidic components on the computer. Based on the commercial CFD solver COMSOL Multiphysics, selected microfluidic components such as micromixers and micro heat exchangers will be simulated and optimized. The tutorial includes a brief introduction to the COMSOL Multiphysics software. Familiarity with basic concepts of Computational Fluid Dynamics is a prerequisite. The course is limited to a maximum of seven students and will take place from April 10 to April 18, 2019.

