CFD simulation and experimental validation of slug flow formation in a microfluidic T-junction device

Tutors: Agnieszka Ladosz, ML H16, 044 632 43 17, aladosz@ipe.mavt.ethz.ch

Type: experimental 30%, theoretical 70%

Background
Microfluidics is a rapidly developing field which has been utilized to realize entire chemical processes in channels of several hundred micrometers. In such environment immiscible fluids form various flow patterns characterised by very high surface to volume ratios and excellent pattern repeatability. This in turns offers such advantages as enhanced heat and mass transfer and reduced reagent consumption when compared to multiphase processes on macroscale. Despite rapid advancements in the field, design of microfluidic devices is still based on trial and error, with the success of the device often depending on the skill and experience of its user. In this aspect, CFD seems as a very promising tool which could help to design microfluidic devices fit for a specific purpose and thus shorten the device development process. Moreover, data which is not easily available experimentally can be easily obtained from a successful CFD simulation and thus can aid the understanding of processes occurring on microscale.

Goals
The goal of this master thesis is to conduct CFD simulations of slug flow formation in a T-junction using commercially available CFD software Fluent and to prepare the experimental database for the validation of performed simulations. For this purpose, student will familiarize himself with the previous studies performed in this area at our institute and based on a literature research, will propose a most suitable approach to continue the work. At the same time he will characterize slug flow experimentally by recording videos of slug flow formation in a PDMS device under various flow conditions using a high-speed camera. Slug sizes and velocities should be read from recorded images using self-developed Matlab code. If time allows, experimental data will be used as a verification for the CFD simulations.

Figure 1. Droplet formation mechanism observed experimentally and in CFD simulations