General instructions You will use a simple MPC program to do two-dimensional simulation runs, where you will study the behavior of the fluid in a microchannel geometry. This means that there are periodic boundary conditions in x-direction and walls in y-direction. The fluid feels a constant force, "gravity", pulling in in the positive x-direction. In exercise 2, there will also be an immobile obstacle.

The simulation program can be compiled as `g++ -o simulation simulation.c`, and subsequently run as `./simulation`. The parameters are read in from the file `parameter.ini`. In the exercises, you will mostly edit this file, but also the source code. You can use whatever text editor you wish. If you need help with editing, please ask.

The simulation results are visualized with gnuplot. To start gnuplot, type `gnuplot`. From gnuplot, you can exit to shell by typing `exit`. There are prepared gnuplot files for visualization of the flow field (`load 'flowfield.gp'`) and the flow profile (`load 'flowprofile.gp'`). If you change the properties of the system (channel size, obstacle, etc.) in the input file, you will have to edit these files as well so that the visualization follows suit.

1 Use the code in directory `exercise1/`. It might be a good idea to copy the files to a working directory so that a pristine copy to revert to always remains.

(a) Compile the code and run it. Plot the flow field and the flow profile. Is the latter parabolic?

(b) Experiment on changing the equilibration time and the total simulation time. What happens?

(c) Measure the velocity profile for different fluid densities, time steps, and applied external forces. When do and when do not the measured and theoretical profiles agree? Why?

(d) Find a way to switch off the grid shift (you will have to edit the source code and recompile for this). What happens?