SPH fluid dynamics model

Kajetan Niewczas

Faculty of Physics and Astronomy, University of Wrocław
kajetan.niewczas@gmail.com

1 The Objective

The main goal of the project was the proper implementation of the Smooth Particle Hydrodynamics model. Apart from simple tests, the main attempt was to recover proper hydrodynamics of the Poiseuille flow and the driven cavity.

2 The SPH Model

2.1 Essentials

Smooth Particle Hydrodynamics is a particle-based Lagrangian method and thus, it introduces particles to define the simulated fluid and to evaluate the arising dynamics properly. These so called smoothed particles, represented as point masses in the simulation domain Ω, carry information like position and velocity as well as certain densities and force fields that describe the fluid's properties. As the name of the method already reveals, the properties of a fluid particle are smoothed out over the neighbouring particles within a support region with radius h.

The integral interpolant of any continuous field quantity $\mathbf{A}$ is,

$$\mathbf{A}_I(\mathbf{x}) = \int_\Omega \mathbf{A}(\mathbf{x}')W(\mathbf{x}-\mathbf{x}', h)d\mathbf{x}, \tag{1}$$

where $\mathbf{x}$ is the position vector in Ω and W is a certain smoothing kernel function with the parameter h as smoothing length.

The numerical equivalent to (1) yields

$$\mathbf{A}_S(\mathbf{x}) = \sum_j \mathbf{A}_j V_j W(\mathbf{x}-\mathbf{x}_j, h), \tag{2}$$

where the integral is approximated by a summation interpolant and the j iterates over every single particle. V_j represents the theoretical volume of a single particle j.

Using this formalism, for each particle, we can calculate density and pressure forces, as well as viscosity or surface tension.

2.2 Basic Model

The first approach was to implement a simplified model with pressure, viscous and gravity forces. The particles propagation was calculated using the simplest Euler method. The boundary conditions were hard walls with the inelasticity parameter.

The results can be found in a simple visualisation of 2000 particles forming a drop falling down: "drop_grav_2000.gif".

2.3 Boundary Conditions

The next improvement was a better implementation of the no-slip boundary conditions. It has been performed using the "mirror particles" method. For each particle, that kernel range overlapped a wall, is construced a mirror particle on the opposite side, with the same density, but opposite velocities. This method allows to keep the proper density near the boundaries and keeps the no-slip conditions.

3 Simulations

3.1 The Poiseuille Flow

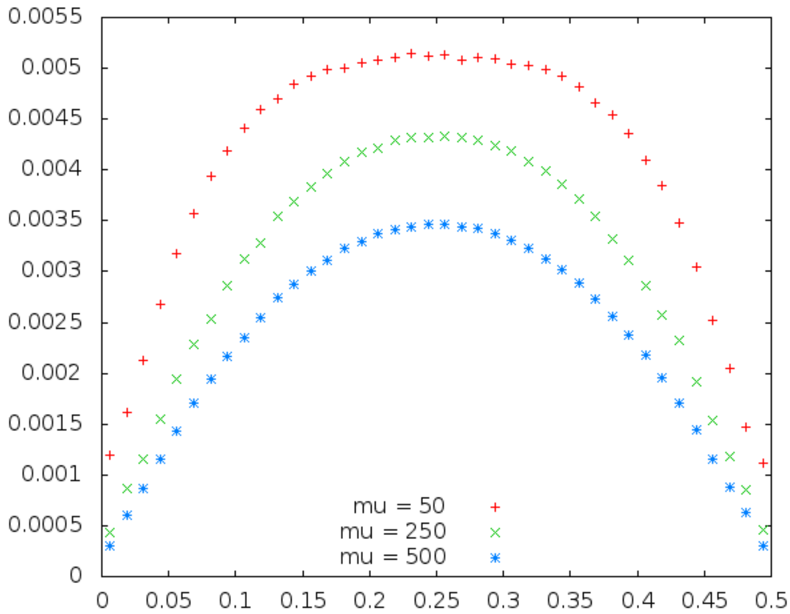

Figure 1: Poseuille flow velocity profiles

The simulation of the Poiseuille flow has been performed for three different viscosities ($\mu_1 = 50$, $\mu_2 = 250$, $\mu_3 = 500$), all with satisfactory results. The velocity profiles for μ_2 and μ_3 have a proper parabolical shape. For μ_1 the simulated pipe was too wide. The visualization can be found in files: "poiseuille_mu50.gif", "poiseuille_mu250.gif" and "poiseuille_mu500.gif" (the last one with particles colored according to their velocity).

3.2 The Driven Cavity

The second performed test was the driven cavity. Apart from some difficulties with a proper implementation of the solid box boundary conditions, the test resulted well. The visualization can be found in a file "driven_cavity.gif".

4 Summary

The basic SPH model has been implemented. It preserves proper hydrodynamics, as it passed a common test: the Poiseuille flow. It is a good base for further improvement. Some of the things considered are:

- Visualisation (OpenGL, Marching cubes algorithm, Real-time),
- Generic method of setting the simulation (Inlet/Outlet, Boundaries),
- Code optimalization (Neighbour lists).