1 The SPH equations

Smoothed particle hydrodynamics (SPH) is a particle-based method for simulating the behavior of fluids. Each computational particle carries along information about the fluid in a little region, such as the velocity and density; and during the course of the simulation, these particles interact with each other in a way that models the dynamics of a fluid. In this project, we will tune a simple 3D SPH method described by Müller et al for use in graphics [1]. There are better methods for this problem (and this implementation is arguably incomplete – we left out the surface tension forces), but this method does illustrate common issues in particle-based methods.

Our simulation basically solves a system of ordinary differential equations[1] for a collection of particles with equal masses m and interaction radii h. Each particle i has a position $\mathbf{r}_i$, a velocity $\mathbf{v}_i$, and a density ρ_i. Particle i interacts with the set N_i of particles within radius h of i. The density is computed at each step by

$$\rho_i = \frac{4m}{\pi h^8} \sum_{j \in N_i} (h^2 - r^2)^3.$$

The acceleration is computed by the rule

$$\mathbf{a}_i = \frac{1}{\rho_i} \sum_{j \in N_i} \mathbf{f}_{ij}^{\text{interact}} + \mathbf{g},$$

where

$$\mathbf{f}_{ij}^{\text{interact}} = \frac{45}{\pi h^5} \frac{m_j}{\rho_j} (1 - q_{ij}) \left[\frac{k}{2} (\rho_i + \rho_j - 2\rho_0) \frac{(1 - q_{ij})}{q_{ij}} \mathbf{r}_{ij} - \mu \mathbf{v}_{ij} \right],$$

where $\mathbf{r}_{ij} = \mathbf{r}_i - \mathbf{r}_j$, $\mathbf{v}_{ij} = \mathbf{v}_i - \mathbf{v}_j$, and $q_{ij} = \|\mathbf{r}_{ij}\|/h$. The parameters in these expressions are

$$\begin{aligned}
\rho_0 &= \text{reference mass density} \\
k &= \text{bulk modulus} \\
\mu &= \text{viscosity} \\
\mathbf{g} &= \text{gravitational vector}
\end{aligned}$$

By default, we choose most of these parameters to be appropriate to a liquid like water. The exception is the bulk modulus, which is chosen so that the computational speed of sound

$$c_s = \sqrt{\frac{k}{\rho_0}}$$

is large relative to the typical velocities we expect to see in the simulation, but not too large. Choosing k to be very large (e.g. on the scale of the bulk modulus for water) severely limits the time step size needed for stable simulation.

[1] We describe the derivation of the equations in a separate document. It may interest those of you who care about fluid dynamics, but it is not critical to understand the derivation in order to do the assignment.

References

[1] M. Müller, D. Charypar, and M. Gross. *Particle-based fluid simulation for interactive applications*, in Proceedings of Eurographics/SIGGRAPH Symposium on Computer Animation.