Smoothed Particle Hydrodynamics and its parallel implementation

Zhenyu He Louis F. Rossi

Mathematical Sciences, University of Delaware – Newark DE, USA

Introduction

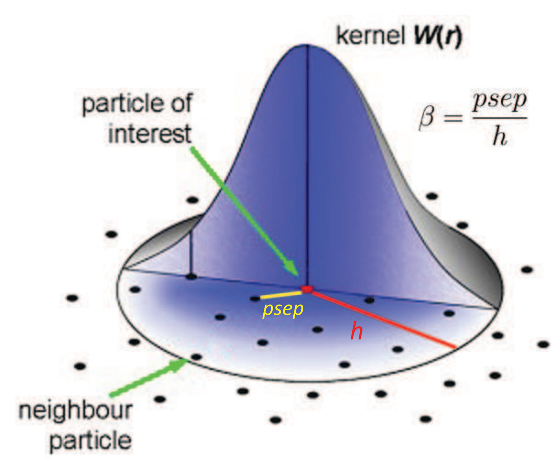
Smoothed particle hydrodynamics (SPH) is a numerical method for obtaining approximate solutions of the equations of the fluid dynamics by replacing the fluid with a set of particles. From a mathematical point of view, these particles are interpolation points from which the properties of the fluid are interpolated by moving basis functions. The mesh-free formulation of the method and its inherent stability make it popular for problems that have complex geometry or large deformations.

Essential formulation of SPH

SPH was developed to solve fluid dynamic problems in forms of a system of partial differential equations (PDEs). In the system, the rates of change of physical quantities depend on the spatial derivative of physical quantities. SPH approximates these derivatives using the information of a finite number of moving particles, by interpolating on these particles with the smoothing kernel function.
SPH interpolation of a quantity f, is based on integral interpolant

$$f_I(\mathbf{r}) = \int f(\mathbf{r}')W(\mathbf{r}-\mathbf{r}', h)\,d\mathbf{r}'$$

where the function W is the smoothing kernel and $d\mathbf{r}'$ is the volume element. Kernel functions are well chosen, normalized functions which have compact support, and tend to the delta function as the length scale h tends to zero. Three key parameters of kernel functions are shape, width h and overlap factor β.



We approximate the integral and the first derivative of f as a summation over the mass element

$$f_S(\mathbf{r}) = \sum_j f_j W(\mathbf{r}-\mathbf{r}_j, h)\frac{m_j}{\rho_j} \qquad \nabla f_S(\mathbf{r}) = \sum_j f_j \nabla W(\mathbf{r}-\mathbf{r}_j, h)\frac{m_j}{\rho_j}$$

Since W falls rapidly with distance, this summation is over only neighboring particles in a local domain.

Governing Equations of Fluid dynamics and SPH formulation

The governing equations for dynamic fluid flows can be written as a set of partial differential equations in lagrangian description.
Continuity equation:

$$\frac{\mathrm{d}\rho}{\mathrm{d}t} = -\rho\nabla\cdot\mathbf{v}$$

Momentum equation in absence of external force:

$$\frac{\mathrm{d}\mathbf{v}}{\mathrm{d}t} = -\frac{1}{\rho}\nabla P + \frac{1}{\rho}\nabla\cdot(\mu\nabla\mathbf{v})$$

Using the kernel estimate and particle approximation, one gets,

$$\frac{d\rho_i}{dt} = \sum_j m_j \mathbf{v}_{ij}\nabla_i W(\mathbf{r}_{ij}, h)$$

where $\mathbf{v}_{ij} = \mathbf{v}_i - \mathbf{v}_j$ and $\mathbf{r}_{ij} = \mathbf{r}_i - \mathbf{r}_j$.
For the momentum equation, in order to conserve momentum, the expression is modified for the term $\frac{\nabla P}{\rho}$

$$\frac{\nabla P}{\rho} = \nabla\left(\frac{P}{\rho}\right) + \frac{P}{\rho^2}\nabla\rho$$

By applying the SPH derivative formulation to the right hand side,

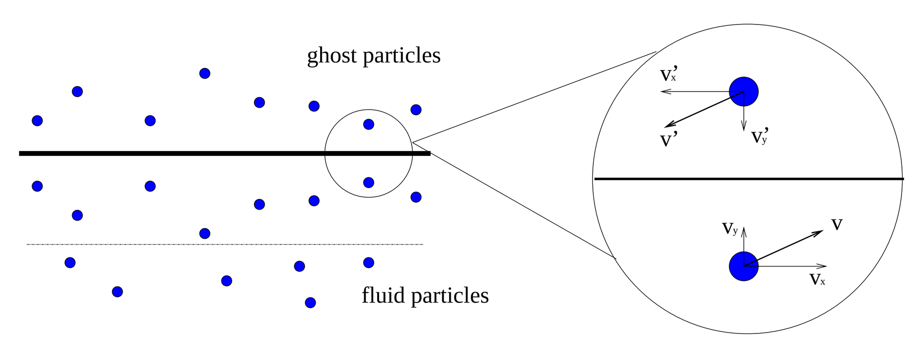
$$\frac{d\mathbf{v}_i}{dt} = -\sum_j m_j\left(\frac{P_i}{\rho_i^2} + \frac{P_j}{\rho_j^2} + \Pi_{ij}\right)\nabla_i W(\mathbf{r}_{ij}, h)$$

where Π_{ij} is the viscous diffusion term. This comes from a hybrid expression combining a SPH first derivative with a finite difference approximation of a first derivative,

$$\Pi_{ij} = \frac{\mu_i + \mu_j}{\rho_i\rho_j}\frac{\mathbf{v}_{ij}\cdot\mathbf{r}_{ij}}{\mathbf{r}_{ij}^2 + 0.01h^2}$$

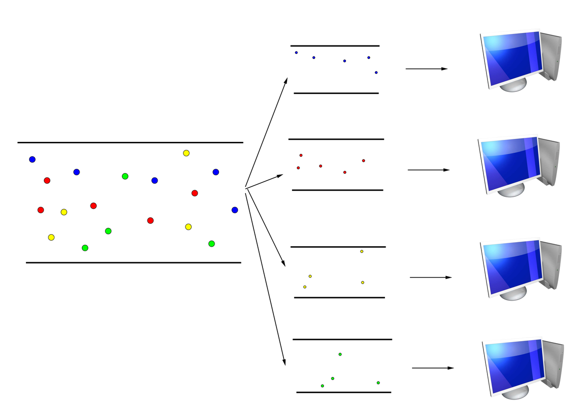
Boundary treatment

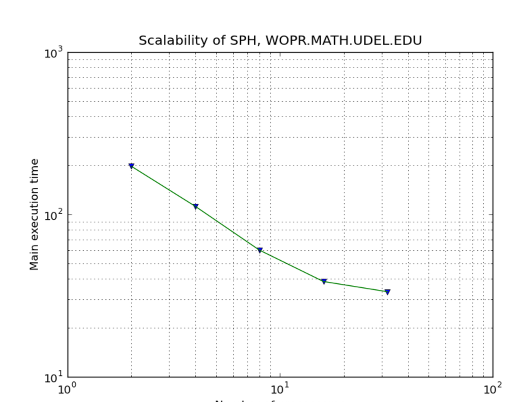
Special technique are used to capture real physical boundary conditions. Ghost particles are created outside the fluid domain by reflecting fluid particles across the boundary. They have the same density, mass and pressure as corresponding fluid particles, but with the perpendicular component of the velocity having the opposite sign to achieve no-penetration condition, and the tangential component having the opposite sign to achieve no-slip condition.



Parallel implementation

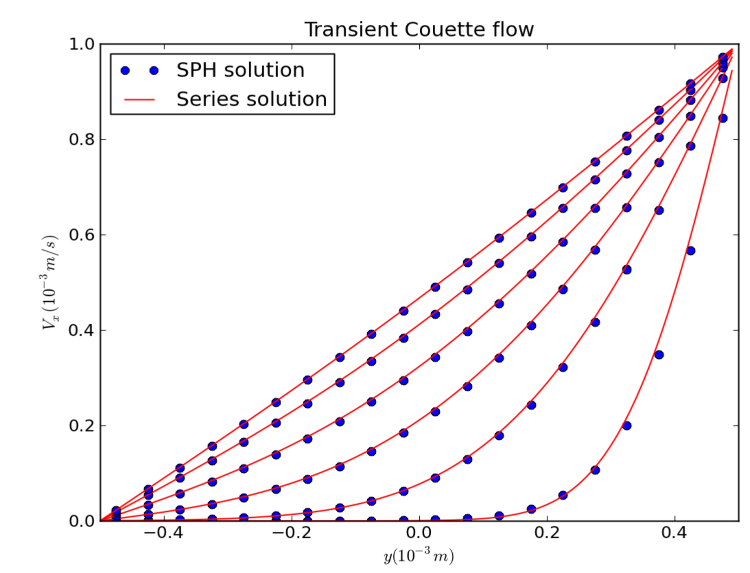
Since SPH is a system of locally interacting particles, it is a perfect candidate for parallelization. Based on particle decomposition, a group of particles are assigned to a particular processor during the simulation, irrespective of its spatial location.



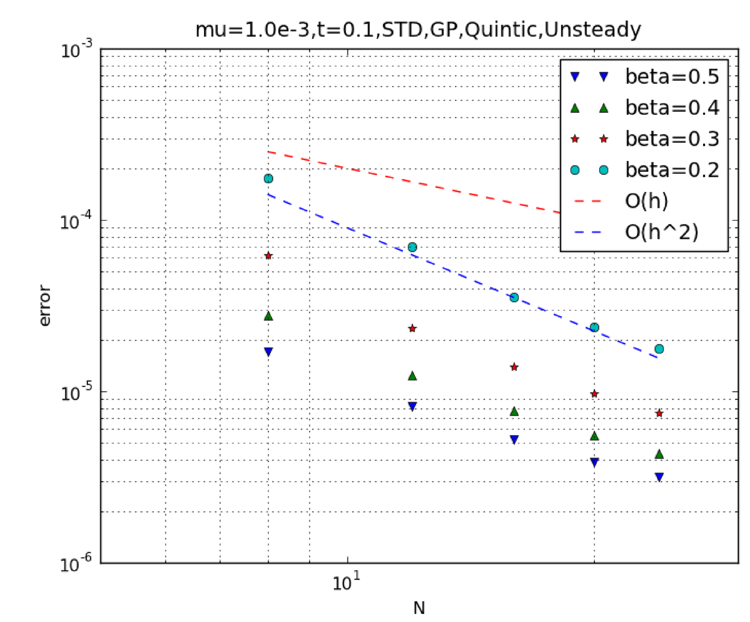


Simulation of the Couette flow

The Couette flow is a fluid flow between two infinite plates located at $y = -L$ and $y = L$. The flow is generated after the upper plate moves at constant velocity U parallel to the x-axis. The series solution for the time-dependent behavior is known. In the figure, we compare the SPH solution and the exact series solution.



The accuracy of SPH is related to many factors, such as kernel function shape, time integration, boundary condition treatment, number of particles, and overlap factor, etc. The figure below shows how the error depends on number of particles N and overlap factor β. It shows that in our case SPH method has second order accuracy.



Conclusion and future work

The ghost particles give good results for flows with simple geometries. It is not clear how the ghost particles should be placed when the boundary has complex geometry. An efficient and general technique needs to be developed. On the other hand, in SPH, the dynamics of a material is governed by the local influence of neighboring particles. Therefore, the efficient querying and processing of particle neighbors is crucial for the performance of the simulation. Our naive parallel implementation works well for low core counts, but not as well for high core counts. Future work will also focus on improvements to our parallel algorithm.

Monaghan, J. J. (2005). Smoothed particle hydrodynamics. Reports on Progress in Physics, 68(8), 1703-1759.
Morris, J. P., Fox, P. J., & Zhu, Y. (1997). Modeling Low Reynolds Number Incompressible Flows Using SPH. Journal of Computational Physics, 136, 214-226.