Numerical simulation of fluid flow in a channel using smoothed particle hydrodynamics

Justin Antony and Ranjith Maniyeri*

Biophysics Laboratory, Department of Mechanical Engineering, National Institute of Technology Karnataka (NITK), Surathkal, Mangalore-575025, Karnataka, India.

ABSTRACT

Smoothed particle hydrodynamics (SPH) is a meshless particle based Lagrangian method invented for modeling astrophysical problems and later extended widely for solving solid and fluid mechanics problems. In meshfree particle methods, arbitrarily distributed particles are used to discretize the problem domain. The arbitrarily distributed particles do not require a fixed connectivity as in grid based methods, thus treatment of large deformations becomes much easier. In addition, discretization of complex domains is much easier compared to the grid based methods as only an initial particle discretization is required. Unlike other meshfree methods where the meshfree nodes only act as interpolation points, SPH has an attractive feature that the nodes also function as material component by carrying properties of the material and move according to the internal and external interactions. In SPH, a smoothing kernel function is used to approximate the field variables and its derivatives at a particle from its neighboring particles. In this paper, we present numerical model developed using SPH to simulate fluid flow through a channel. We consider three cases of fluid flow - Poiseuille flow, shear flow and Couette flow in the channel. Through numerical simulations, it is found that our developed model in FORTRAN matches very well with available analytical results. We believe that this model can be easily extended to complex fluid dynamic problems that involve fluid-structure interactions.

Keywords: Couette flow, Meshfree particle method, Poiseuille flow, Shear flow, Smoothed particle hydrodynamics

1. INTRODUCTION

Smoothed particle hydrodynamics (SPH) was originally invented for modeling astrophysical problems in three-dimensional open space [1]. Later SPH has been successfully implemented for modeling incompressible flows, multi-phase flows, flow through porous media, heat transfer and mass flow, shock simulations, fracture of brittle solids, metal forming and explosion problems [2]. SPH being a meshless method has several advantages over grid-based methods especially in problems with large deformations and complex geometries. However, the conventional SPH has a low accuracy and it cannot exactly reproduce even a constant function. Particle inconsistency is the main problem causing this inaccuracy [3]. Several approaches have been proposed to restore the particle consistency. One way is reconstructing the smoothing kernel to satisfy discretized consistency conditions as in reproduced kernel particle method (RKPM) proposed by Liu et al. [4]. Another way is to construct improved SPH approximation schemes using Taylor series expansions. Corrective smoothed particle method (CSPM) proposed by Chen et al. [3], finite particle hydrodynamics (FPM) proposed by Liu et al. [5] and decoupled finite particle method (DFPM) proposed by Zhang and Liu [6] are schemes using Taylor series expansions. The CSPM and FPM schemes

*Corresponding author: mranji1@nitk.edu.in

calculates the field variables and its derivatives by inverting a matrix, which may lead to numerical instability or unexpected termination of simulation for highly disordered particle distributions. In DFPM, there is no need to solve matrix equations thus it is flexible, stable and computationally more efficient. However, it uses a nested summation to approximate the higher order derivatives from lower order derivatives and this increases computational time. The symmetrized SPH equations reduced errors due to the particle inconsistency problems [2]. Morris et al. [7] reported a simplified SPH equation for viscous terms in momentum equation that exactly conserves the linear momentum and approximately conserves angular momentum.

In this paper, we present a numerical model developed using SPH in FORTRAN to solve the continuity and momentum equations to simulate the fluid flow between two parallel plates. We have used symmetrized SPH equations [2] for pressure gradient term and simplified SPH equation for viscous terms [7]. CSPM [3] (or DFPM, both are same in the case of evaluation of function at a point) is used for evaluating field variables of virtual particles that have been used to improve the accuracy of particles near boundary.

2. METHODOLOGY

The SPH method consists of two steps. The first step is to discretize the problem domain into a number arbitrarily distributed interpolation points. The second step is the formulation of SPH equations. In SPH, a smoothing kernel function is used to approximate field variables and its derivatives at a particle through an interpolation over the neighboring particles. The field variables are first expressed in the form of integral representations using the kernel function and then the integration over the support domain is replaced with summation over the neighboring particles. The SPH equations for a function and its derivative are given in Eqs. (1) and (2).

$$< f(\mathbf{x}_i) >= \sum_{j=1}^{N} \frac{m_j}{\rho_j} f(\mathbf{x}_j) \cdot W_{ij} \tag{1}$$

$$< \nabla \cdot f(\mathbf{x}_i) >= \sum_{j=1}^{N} \frac{m_j}{\rho_j} f(\mathbf{x}_j) \cdot \nabla_i W_{ij} \tag{2}$$

where i and j represents each particles, N is the number of neighboring particles in the support domain of particle i, m is the mass, ρ is the density, $W_{ij} = W(\mathbf{x} - \mathbf{x}', h)$ is the smoothing kernel function, $\nabla_i W_{ij} = \frac{x_j - x_i}{r_{ij}} \frac{\partial W_{ij}}{\partial r_{ij}}$ and **x** is the position vector. Here, h is the smoothing length that defines the support domain for a particle and only those particles within the support domain of the interested particle will contribute to calculate the field variables at that particle. The angled bracket is used to denote that it is an approximation using smoothing kernel function.

The SPH equations for the continuity and momentum are represented in Eqs. (3) and (4) as

$$\frac{\partial \rho_i}{\partial t} = \rho_i \sum_{j=1}^{N} \frac{m_j}{\rho_j} \mathbf{v}_{ij} \nabla_i W_{ij} \tag{3}$$

$$\frac{d\mathbf{v}_i}{dt} = \sum_{j=1}^{N} \frac{m_j}{\rho_j} \left(\frac{P_i}{\rho_i^2} + \frac{P_j}{\rho_j^2} \right) \nabla_i W_{ij} + \sum_{j=1}^{M} m_j \frac{\left(\mu_i + \mu_j\right) \mathbf{v}_{ij}}{\rho_i \rho_j} \left(\frac{1}{r_{ij}} \frac{\partial W_{ij}}{\partial r_i} \right) + F_i \tag{4}$$

where ρ, m, v, P and μ are density, mass, velocity, pressure and viscosity of the fluid respectively. $v_{ij} = v_j - v_i$ and F denotes the body force.

The steps followed in our computational model starts from the initial discretization of the domain with real and virtual particles and assigning them initial properties. In this work, we have used uniformly distributed particles and all particles are considered to be at rest initially. The neighbouring particles of each particle are estimated using all-pair search algorithm [2] and stored them in a two-dimensional array. Then, the properties of the virtual particles are determined from the real fluid particles using CSPM to impart the boundary conditions and to improve the accuracy of near boundary particles. The virtual particles are the dummy particles that placed outside the boundaries to complete the support domain of particles near boundary. These particles are fixed in position but have artificial velocities to only impose boundary conditions. After updating the properties of virtual particles, the momentum equation is solved for all the real fluid particles and their velocities are calculated. Then the positions of the real fluid particles are updated according to their velocity using a simple forward time stepping. As the particles move, the local densities will vary and continuity equation is solved to update the densities of real fluid particles. Finally, an artificial equation of state [8] is used to update the pressure .The artificial equation of state, $P(\rho) = c^2(\rho - \rho_0)$ is an explicit function of density, which is used to include the incompressibility condition so that the density variations are within 1%. The all-pair search algorithm is used to update the neighbours of each particle at the beginning of next time step and the procedure continues till the specified time is reached.

3. RESULTS AND DISCUSSION

The fluid flow between two infinitely long parallel plates is modeled with a domain of size 5.0×10^{-4} m × 1.0×10^{-3} m using periodic boundary condition in flow direction. In periodic boundary condition, the fluid leaving one side reenters though the opposite side and the particles at the inlet side interacts with particles at the outlet side. The domain is discretized with 40×80 uniformly distributed particles. Additional four layers of particles (virtual particles) added above and below the top and bottom walls respectively, to avoid truncation of support domain of the particles near the wall. Poiseuille flow, Couette flow and shear flow have been simulated by varying the boundary conditions imposed on the parallel plates. The time step is chosen as 1.0×10^{-5}s, using the stability criteria of viscous diffusion [7] and a smoothing length of 1.1 times initial particle spacing is adopted in all simulations.

3.1 Poiseuille Flow

We develop a two-dimensional computational model using smoothed particle hydrodynamics based method and a code is built in FORTRAN. Using the developed model, first of all we simulate Poiseuille flow, in which both the parallel plates are kept fixed and the fluid flows through the channel by means of an external body force of magnitude 8.0×10^{-4}**,** which corresponds to Reynolds number (Re) 0.1. The same case has been simulated for a different

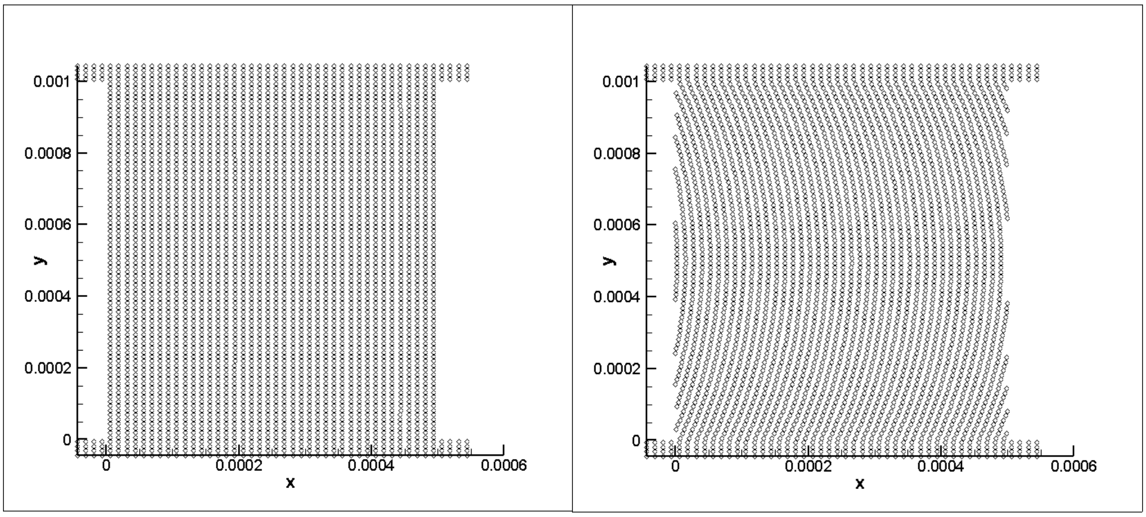

Figure 1. Initial and final particle distributions for Poiseuille flow

Reynolds number (Re=0.05) by reducing the height of the channel to half of the initial height with 20×40 particles and keeping all other parameters same. The obtained results are compared with the analytical results.

Morris et al. [7] provided the analytical solution of the velocity profile for Poiseuille flow. Figure 1 shows the initial and final particle distributions of Poiseuille flow for Re = 0.1.

Figure 2 shows the comparison of results obtained using SPH and analytical results of Poiseuille flow for both Reynolds numbers. The numerical results are in close agreement with the analytical results.

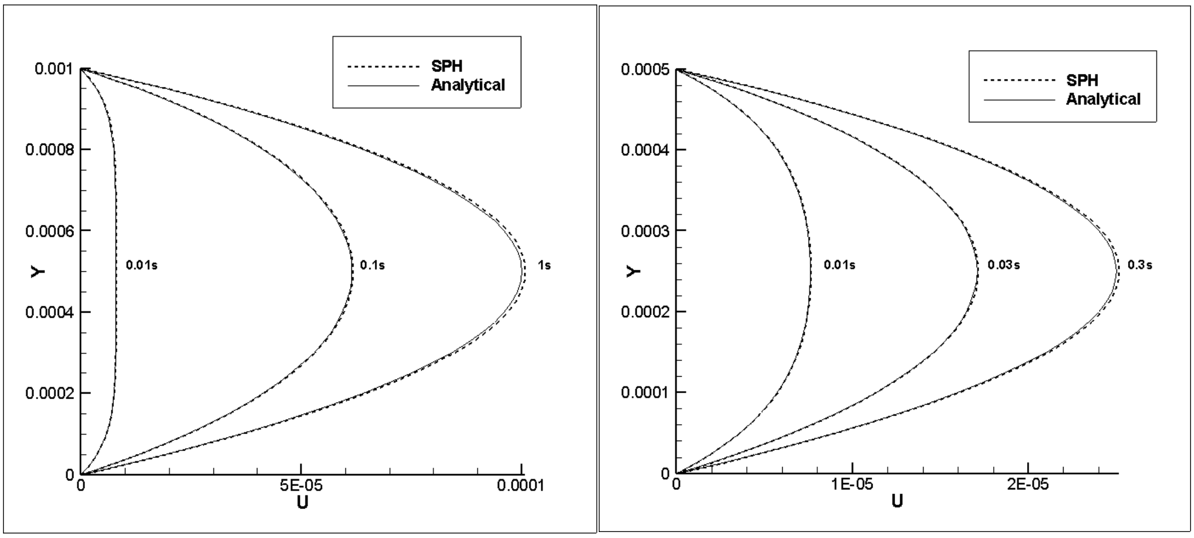

Figure 2. Comparison of velocity profiles obtained from SPH and analytical results for Poiseuille flow with Re=0.1 (left) and Re=0.05(right).

3.2 Couette Flow

In this case, fluid flows between two infinite parallel plates with one plate (bottom) fixed and the other plate (upper plate) moving with a specified velocity (V_0) of $1.0 \times 10^{-4} \mathrm{m/s}$ (corresponds to Re=0.1. All channel parameters are same as that of initial case of Poiseuille flow. In this case, no external force is applied, the fluid flow is solely due to the movement of the upper plate. Figure 3 shows the comparison of results obtained from our computational model and the analytical results by Morris et al. [7]. It is found that they are in close agreement with maximum percentage error less than 0.02.

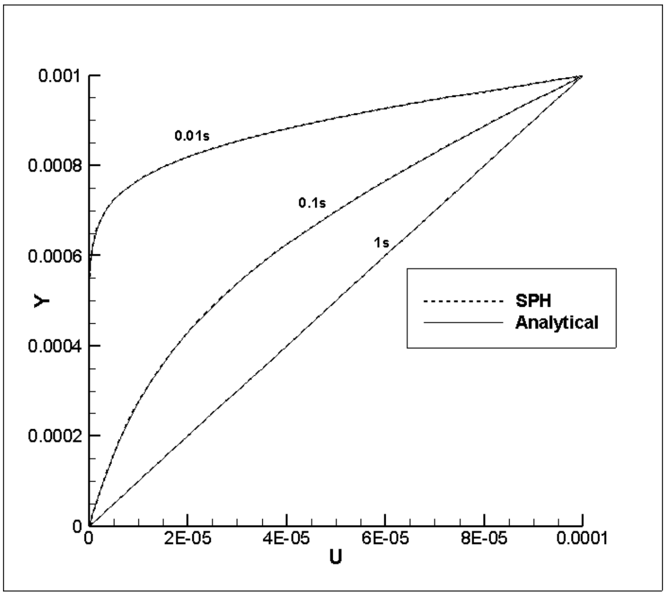

Figure 3. Comparison of velocity profiles obtained from SPH and analytical results of Couette flow with Re=0.1

3.3 Shear Flow

Finally, a case of shear flow has been simulated with the same domain but with both walls moving in opposite direction with same velocity of $1.0 \times 10^{-4} m/s$ (Re=0.1). All other parameters are same as that of initial case and fluid flow was only due to the movement of walls. The SPH was able to exactly reproduce the linear steady state velocity profile. The velocity profiles at 0.01s and 1s is shown in fig.4.

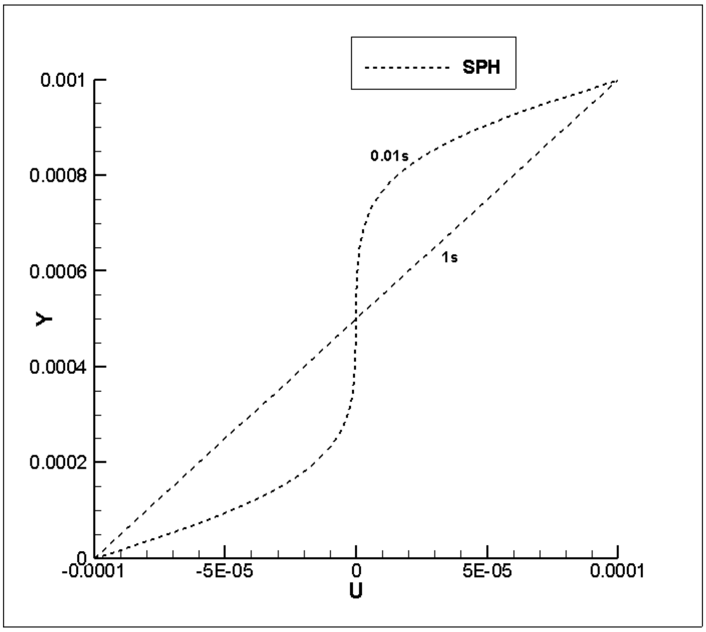

Figure 4. Velocity profiles obtained from SPH for shear flow with Re=0.1

4. CONCLUSION

In this paper, we developed a computational model using smoothed particle hydrodynamics (SPH) to solve continuity and momentum equations to simulate three types of fluid flow in a channel. The symmetrized SPH equation is used for pressure gradient term in momentum equation and the viscous term in momentum equation is calculated using modified equation of Morris et al. [7]. CSPM is used for calculating the field variables of virtual particles that are used to improve the accuracy of particles near boundaries. The developed model in SPH is used to simulate Poiseuille flow, Couette flow and shear flow in channel. The numerical results obtained are compared with available analytical results and found to be in good agreement. In near future, we aim to extend this model to study fluid-structure interaction problems in the field of biofluid dynamics.

REFERENCES

1. R. A. Gingold and J. J. Monaghan, "Smoothed particle hydrodynamics: theory and application to non-spherical stars," Monthly notices of the royal astronomical society 181 (3), pp. 375-389, (1977).
2. G. R. Liu and M. B. Liu, “Smoothed particle hydrodynamics: a meshfree particle method”. World scientific, (2003).
3. J. K. Chen, J. E. Beraun and T. C. Carney,” A corrective smoothed particle method for boundary value problems in heat conduction”, Int. J. Numer. Meth. Engng. 46, pp. 231-252, (1999).
4. W. K. Liu, Y. Chen, S. Jun, J. S. Chen, T. Belytschko, C. Pan, R. A. Uras and C. T. Chang, “Overview and applications of the reproducing Kernel particle methods”, Arch. Comput. Methods Eng., 3, pp. 3–80, (1996).
5. M. B. Liu, W. P. Xie and G. R. Liu,” Modeling incompressible flows using a finite particle method”, Applied Mathematical Modelling, 29, pp. 1252–1270, (2005).
6. Z. L. Zhang and M. B. Liu, "A decoupled finite particle method for modeling incompressible flows with free surfaces." Applied Mathematical Modelling 60 pp. 606-633, (2018).
7. J. P. Morris, P. J. Fox and Y. Zhu, "Modeling low Reynolds number incompressible flows using SPH." Journal of computational physics 136 (1), pp. 214-226, (1997).
8. J. Fang, R. G.Owens, L. Tacher, A. Parriaux, “A numerical study of the SPH method for simulating transient viscoelastic free surface flows”, J. Newt. Fluid Mech., 139(1), pp. 68-84, (2006).