SMOOTHED PARTICLE HYDRODYNAMICS SIMULATION OF BLOOD FLOW IN THE LEFT VENTRICLE IN DIASTOLIC PHASE

Shahrokh Shahriari , Ibrahim Hassan and Lyes Kadem

Mechanical and Industrial Engineering Department, Concordia University, Montreal, Canada

INTRODUCTION

The main difficulties of grid-based numerical methods come from the use of mesh. The descritized equations in grid-based numerical methods are based on the generation or regeneration of meshes that can be a time-consuming and costly work.

Smoothed Particle Hydrodynamics (SPH) is a meshfree particle method (fully Lagrangian method), originated to model astrophysical phenomena [1, 2], and later developed for applications in continuum solid and fluid mechanics. In SPH method the fluid field is divided to a set of discrete fluid elements (particles). These particles have a spatial distance, over which their properties are "smoothed" by a weighting function. This means that any physical quantity of a particle can be calculated by summing the relevant properties of all the particles within its support domain.

The earliest applications of SPH in fluid dynamic field were in the compressible fluid dynamics area. However, in SPH method incompressible flows can be treated as an inconsiderable compressible flow using an artificial equation of state [3]. SPH method has very wide applications from micro-scale to astronomical scale continuum phenomena. Furthermore, SPH method is very suitable for problems with complex deformable boundaries. These kinds of problems are usually encountered in the human body and more specifically in the cardiovascular system.

The present work is, therefore, a first attempt to simulate the unsteady filling phase (diastolic phase) in a 2D simplified model of a left ventricle using SPH method.

THE SPH METHOD

The main characteristic of the SPH method is based on integral interpolants [4]. In this method, the fundamental principle is to approximate any function $A(\vec{r})$ by,

$$\left\langle A(\vec{r})\right\rangle = \int A(\vec{r}')\,W(\vec{r}-\vec{r}',h)\,d\vec{r}' \tag{1}$$

where h is called the influence or smoothing length and $W(\vec{r}-\vec{r}',h)$ is the weighting function or kernel. It should be normalized, and show a behavior like delta function as the smoothing length approaches to zero, and have continuous first and second derivatives.

In numerical works this approximation leads to the following approximation for any function at an interpolation point “a”,

$$\left\langle A(\vec{r}_a)\right\rangle = \sum_b m_b \frac{A_b}{\rho_b} W(\vec{r}_a-\vec{r}_b,h) \tag{2}$$

The summation is over all the particles within the region of compact support of the kernel function. m_b and ρ_b are mass and density of particle “b” in the compact support area of particle “a”.

The gradient of a function for particle “a” can be obtained from its values at neighboring,

$$\vec{\nabla} A(\vec{r}_a) = \sum_b m_b \frac{A_b}{\rho_b} \vec{\nabla} W(\vec{r}_a-\vec{r}_b,h) \tag{3}$$

In this way, the Navier–Stokes equations can be presented in the form of SPH formulation.

Weighting function (kernel)

The correct selection of the weighting function has an important effect on the performance of SPH modeling. Different kinds of weighting functions are suggested in the literature. The function depends on the smoothing length, h, and the distance between particles “a” and “b”, r_{ab}. The smoothing length determines the size of support domain of particle “a”, where particles in it contribute to the approximation. Most of simulations in SPH employ the following cubic spline kernel [4]:

$$W_{ab} = \frac{10}{7\pi h^2}\begin{cases} 1-\frac{3}{2}\left(\frac{r_{ab}}{h}\right)^2+\frac{3}{4}\left(\frac{r_{ab}}{h}\right)^3 & ;\ (0 \le \frac{r_{ab}}{h} \le 1) \\ \left(2-\frac{r_{ab}}{h}\right)^3/4 & ;\ (1 \le \frac{r_{ab}}{h} \le 2) \\ 0 & ;\ (2 \le \frac{r_{ab}}{h}) \end{cases} \tag{4}$$

Continuity equation

The continuity equation can be written under its SPH formulation as follows [4],

$$\frac{d\rho_a}{dt} = \sum_b m_b \vec{V}_{ab} \vec{\nabla}_a W_{ab} \tag{5}$$

where $\vec{V}_{ab} = \vec{V}_a - \vec{V}_b$.

Momentum equation

The momentum conservation equation for a fluid particle "a" in the field is,

$$\left.\frac{d\vec{V}}{dt}\right|_a = -\frac{1}{\rho}\vec{\nabla}P\Big|_a + \nu\nabla^2\vec{V}\Big|_a \tag{6}$$

where $\vec{\nabla}P$ is the pressure gradient and ν refers to the kinematic viscosity of the fluid.

Pressure gradient could be estimated by using,

$$-\frac{1}{\rho}\vec{\nabla}P\Big|_a = -\sum_b m_b \left(\frac{P_b}{\rho_b^2} + \frac{P_a}{\rho_a^2}\right)\vec{\nabla}_a W_{ab} \tag{7}$$

Viscous stress term is presented in the literature in different ways [5]. We modeled this term as follows,

$$\nu\nabla^2\vec{V}\Big|_a = \sum_b m_b \left(\frac{8\,(\nu_a + \nu_b)\,\vec{V}_{ab}\,\vec{r}_{ab}}{(\rho_a + \rho_b)\left|\vec{r}_{ab}\right|^2}\right)\vec{\nabla}_a W_{ab} \tag{8}$$

where $\vec{r}_{ab} = \vec{r}_a - \vec{r}_b$.

Flow pressure

Incompressible flows in the SPH method are treated as weakly compressible, so it is easy to use an equation of state to determine fluid pressure [5],

$$P = c^2 \rho \tag{9}$$

where c is the speed of sound. The speed of sound should be selected so that density varies with less than 1%.

Moving the particles

Particles are moving according to,

$$\frac{d\vec{r}_a}{dt} = \vec{V}_a \tag{10}$$

where $\vec{r}_a$ denotes the position of particle "a".

Boundary condition

Boundary conditions in SPH method are treated in some specific ways. Indeed, when a particle goes near a rigid wall some important facts should be considered: particles near a wall do not have enough neighboring particles in their support domain, which leads to unrealistic results; the particles should not penetrate the rigid walls and a no-slip condition should be satisfied on the walls. Different ways to treat boundary conditions are presented in the literature to take into account these specific considerations [3, 5].

In this work the rigid wall boundaries are represented by some fixed wall particles. These particles have zero velocity. We treat wall particles at corners so that fluid particles do not feel a non-continuous effect when approaching the corners. Two layers of imaginary particles are positioned outside the domain parallel to the wall boundaries. The velocity of these particles is set so that no-slip boundary condition is satisfied on the wall at each time step. The pressure at the wall and of the imaginary particles is calculated during the computation so that Neumann boundary condition is satisfied. Boundary condition at inlet and outlet of the domain should be considered so that the continuity is satisfied.

Implementation

Here a 2D simplified model of a left ventricle (LV) is used to simulate the diastolic phase. The physical geometry and initial distribution of the particles are shown in figure (1).

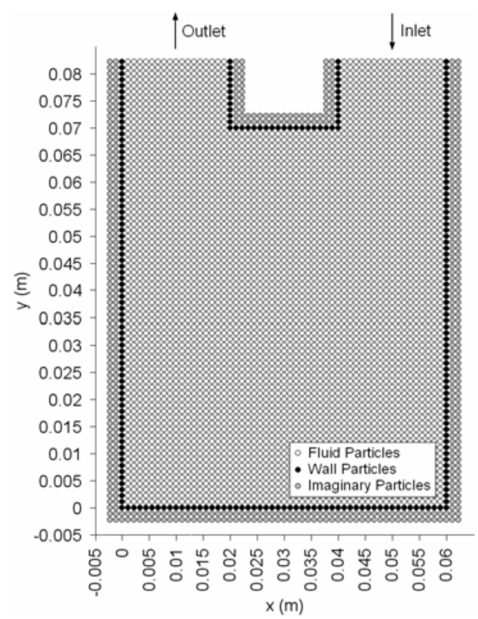

Figure 1: Initial distribution of the particles in the domain.

In the SPH method only particles within the compact support area of a particle are used in the approximation of physical properties. Finding the particles in the support domain of any particle "a" can be performed in different ways. The simplest way is to search the whole computation domain at each time step to check the distance between each particle and particle "a" and to determine if this particle belongs to the support domain of particle "a". Here, we applied the idea of linked list method [6] to search the neighboring particles. In this method some imaginary cells with a dimension equal to the support domain radius are constructed (Figure 2). Then for a particle in a cell only the interaction with the particles in the neighboring cells need to be considered. In this way, the computational time is significantly reduced.

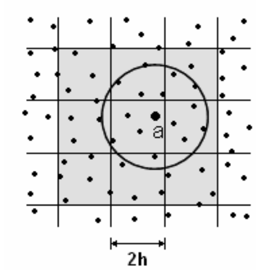

Figure 2: Application of the linked list method.

In this work we compare the effect of applying linked list method instead of using simple neighboring particle search method on the average computation time for one time step. Table (1) shows the results with taking different number of particles. The program has been written in Fortran and runs on an Intel(r) Core (Tm) 2CPU 6300 @ 1.86 GHz computer.

Table 1: Effect of applying linked list method on computation time.

Case	Number of Particles	Computation Time per Time Step	
		Simple Particle Search Method	***Linked List Method***
1	Fluid Particles: 1130 Wall Particles: 140	3 second	0.15 second
2	Fluid Particles: 1633 Wall Particles: 166	6 second	0.25 second
3	Fluid Particles: 2915 Wall Particles: 218	18 second	0.45 second
4	Fluid Particles: 3694 Wall Particles: 244	35 second	0.6 second

The mitral valve inlet velocity is simulated using a flat velocity profile following an unsteady parabolic variation (Figure (3)). Such profile is usually uncounted in patients with atrial fibrillation.

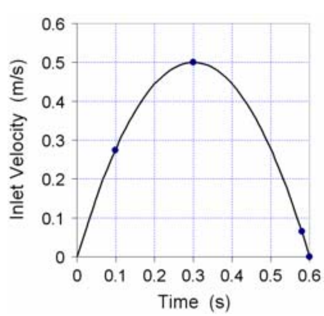

Figure 3: Variation in time of the inlet velocity profile.

The blood flow density and viscosity are set to 1000 kg/m^3 and 3.6×10^{-3} Pa.s.

RESULTS AND DISCUSSION

The simulated velocity field considering 3694 fluid particles with a time step of 1×10^{-5} (s) for four different instants during the cycle are shown in figures (4a-d).

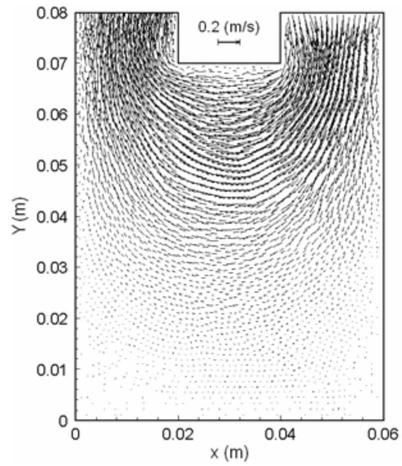

Figure 4a: Velocity field at t=0.1 s.

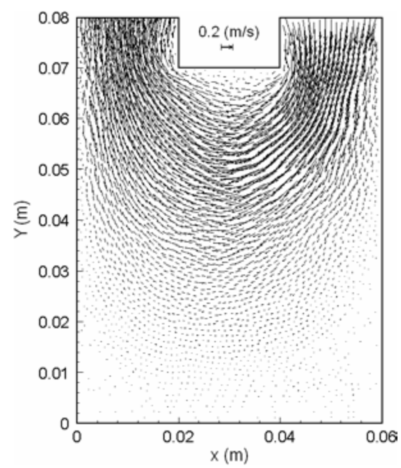

Figure 4b: Velocity field at t=0.3 s.

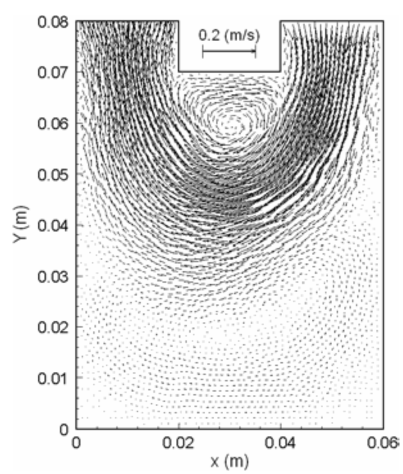

Figure 4c: Velocity field at t=0.58 s.

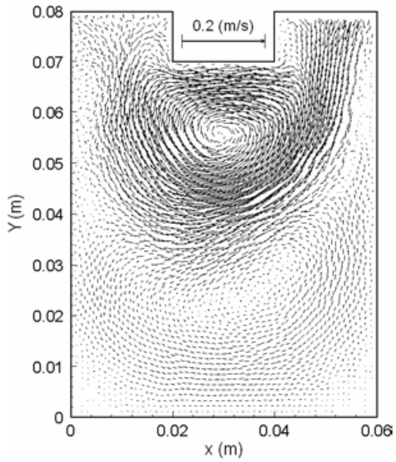

Figure 4d: Velocity field at t=0.6 s.

During the acceleration phase, since in our model the aortic valve is in open position, the flow exiting the mitral valve is immediately directed towards the aortic valve (t=0.1 s). At the peak of the filling phase (t=0.3 s) no significant modification in the flow field can be noticed, except that the jet penetrates more deeply in the cavity (simulating a left ventricle), this can also be noticed on figure (5a-b) showing the spacio-temporal variations in the X and Y components of the velocity.

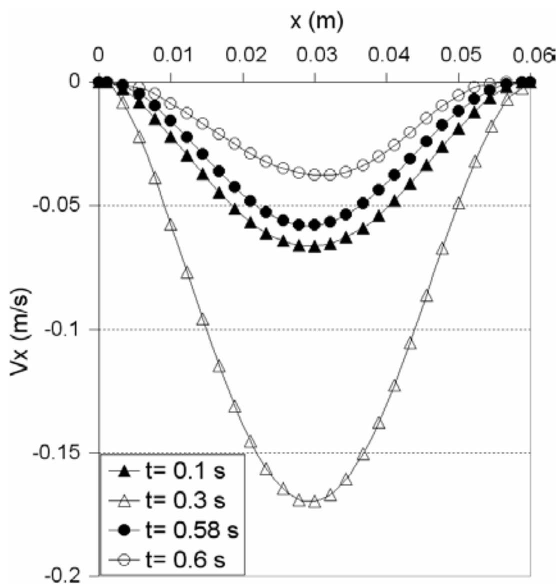

Figure 5a: X- component of the velocity at section y=0.04 m for different instants during the cycle.

During the deceleration phase, as a result of jet breakdown phenomena, a coherent structure appears in the space between the inlet (mitral valve) and the outlet (aortic valve); this coherent structure is then convected towards the centre of the cavity. Although, the simulated conditions are not physiologically correct, the characteristic of the flow, a main vortex structure in the cavity, obtained using SPH method are very close to the one observed in a left ventricle during the filling phase [7].

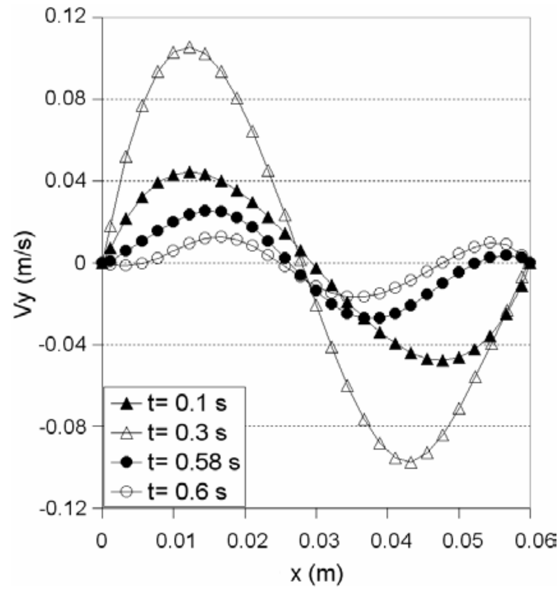

Figure 5b: Y- component of the velocity at section y=0.04 m for different instants during the cycle.

As a conclusion, this work is a first attempt to simulate the filling phase in a realistic left ventricle. To reach physiological conditions, several improvements are still needed, such as using a realistic geometry and taking into account myocardial deformation and the opening and closure of the valves.

REFERENCES

[1] L. B. Lucy, “A numerical approach to the testing of the fission hypothesis”, *Astronomical J.,* vol. 82 (12), pp. 1013-1020, 1977.

[2] R. A. Gingold, and J. J. Monghan, “Smoothed particle hydrodynamics: theory and application to non-spherical stars”, *Monthly Notices of the Royal Astronomical Society*, vol. 181 (2), pp. 375-389, 1977.

[3] J. J. Monaghan, “Simulating free surface flows with SPH”, *J. Computational Physics*, vol. 110, pp. 399-406, 1994.

[4] J. J. Monaghan, “Smoothed particle hydrodynamics”, *Annual Review of Astronomical and Astrophysics*, vol. 30, pp. 543-574, 1992.

[5] J. P. Morris, P. J. Fox, and Y. Zhu, “Modeling low Reynolds number incompressible flows using SPH”, *J. Computational Physics*, vol. 136, pp. 214-226, 1997.

[6] J. C. Simpson, “Numerical techniques for three-dimensional smoothed particle hydrodynamics simulations: applications to accretion disks”, *Astrophysical J.*, vol. 448, pp. 822-831, 1995.

[7] P. J. Kilner, G. Z. Yang, A. J. Wilkes, A. J. Wilkes, R. H. Mohiaddin, D. N. Firmin, and M. H. Yacoub, “Asymmetric redirection of flow through the heart”, *Nature*, vol. 404, pp. 759-761, 2000.