Journal of
Software Engineering

ISSN 1819-4311

Academic
Journals Inc.

www.academicjournals.com

Journal of Software Engineering 8 (4): 321-327, 2014
ISSN 1819-4311 / DOI: 10.3923/jse.2014.321.327

A New Treatment for Boundary of Laminar Flow Inlet or Outlet in SPH

[1]Chaxiang Liang, [1]Jihong Huang and [2]Wei Shi
[1]Department of Computer Science, Wuzhou University, Wuzhou, 543002, China
[2]Department of Mathematics and Physics, Wuzhou University, Wuzhou, 543002, China

Corresponding Author: Chaoxiang Liang, Department of Computer Science, Wuzhou University, No. 82, Fumin 3rd Road, Wuzhou, Guangxi, 543002, China Tel: +86-0774-5825722 Fax: +86-0774-5825722

ABSTRACT

A new method is proposed instead of the periodic boundary conditions in this study which treats the inlet or outlet boundaries with virtual following particles in the Smoothed Particle Hydrodynamics (SPH). The virtual following particles are those located outside the inlet or outlet boundaries and their velocities and positions are updated according to the corresponding interior particles. This method bases on the analysis of characteristics of laminar flow and is able to be applied in low Reynolds number flow which is laminar flow in the inlet or/and outlet boundaries. The Poiseuille flow and two-dimensional flow between two inclined plates are simulated, respectively with the proposed method and the results are in good agreement with theoretical values.

Key words: Virtual following particle, boundary treatment, laminar flow boundary, smoothed particle hydrodynamics, numerical simulation, inlet or outlet boundary

INTRODUCTION

Smoothed Particle Hydrodynamics (SPH) is a kind of particle-based meshless numerical simulation method. Currently, it has been successfully applied to various fields such as astronomy, continuum solid and hydromechanics, etc.

When SPH is applied to problems with a boundary, it does not only need to treat solid wall boundary and/or free boundary but also needs to treat the inlet and outlet boundaries in case of any. In studying the jet buckling problems, Rafiee *et al.* (2007) treated the jet outlet by means of supplementing particles with fixed velocity. In the numerical simulations for the fluid flow in infinite long straight pipe or infinite long straight plate, the periodic boundary conditions were adopted to treat the inlet and outlet boundaries in SPH (Liu and Liu, 2003), in another example, in the research on flow around a circular cylinder, the inlet and outlet boundaries were also treated by means of periodic boundary conditions (Morris *et al.*, 1997). However, for the flows of actual flow fields, the inlet and outlet cannot apply to periodic boundary condition under many circumstances. For instance, in case of circumstances such as inconsistence between the inlet and outlet of flow field, other methods for treating the inlet and outlet boundaries must be sought. Based on Morris' analysis on the two-dimensional flow field between two parallel plates, this study proposes a method of treating the inlet or outlet of laminar flow boundaries under the circumstance of low Reynolds number and verifies the validness of this method through two test cases which shows that numerical simulation results are in good agreement with theoretical values.

TREATING INLET OR OUTLET BOUNDARIES WITH VIRTUAL FOLLOWING PARTICLES

As mentioned above, the periodic boundary condition shall not apply any more in case of inconsistence between the inlet and outlet of flow field. A method, in which the virtual following particles are used to implement the laminar flow inlet or outlet boundary conditions, is proposed as follows.

Assume the two-dimensional flow field between two parallel plates as shown in Fig. 1. For the circumstance of laminar flow, the component of X axis of the velocity at a certain position at a certain moment can be indicated as (Morris *et al*., 1997) in Eq. 1 and 2:

$$Vx\,(y\,,\,t)=\frac{F}{2\upsilon}y(y-L)+\sum_{n=0}^{\infty}\frac{4FL^2}{\upsilon\pi^3(2n+1)^3}\sin[\frac{\pi y}{L}(2n+1)]\exp[-\frac{(2n+1)^2\pi^2\upsilon}{L^2}t] \tag{1}$$

For laminar flow, there always is:

$$Vy = (y, t) = 0 \tag{2}$$

where, y represents the coordinate variable of (Y) axis, (t) time, (F) body force, (υ) kinetic viscosity, (L) height and (n) integer. For a certain flow field and fluid, F, υ and L are constant. Equation 2 shows that during the flowing of fluid, the Y axis coordinate of the particle stays unchanged. Equation 1 shows that the velocity of this particle at a certain position at a certain moment is only related to its Y axis coordinate. At the same time, the two points with the same Y axis coordinate will have the same horizontal velocity. Based on Eq. 1 and 2, it is clear that for any two points with the same Y axis coordinate in laminar flow, the relative velocity between them are always zero. Therefore, the relative distance between them will always keep the relative distance at the original moment and stay unchanged. So, a method that can be used to simulate the laminar flow inlet or outlet boundaries with the virtual following particles is proposed as follows.

The solid dots in Fig. 1 indicate the interior particles or boundary particles of fluid which is discretized. Given the smoothing length of the particles $h = ad_0$, where, (d_0) is the initial particle spacing and (a) is a certain constant determined in line with the problem. The particles located on the upper and lower boundaries are boundary particles. Given AB line is the inlet boundary. If the inlet boundary is not treated, it will be misunderstood as a free surface. To avoid this problem, at the initial setting, some virtual particles with the evenly spacing of d_0 are arranged in parallel with the interior particles on AB line, so as to ensure that the distance between the interior particles on AB line and the outermost virtual particles is not less than $2\tau h$, where, τ is the scaling factor which determines the spread of the specified smoothing function. In the diagram, the virtual particles are indicated by hollow dots to simulate the fluid outside the inlet boundary and implement the boundary condition of laminar flow inlet.

In the same line, the interior particles closest to virtual particles are referred to as the corresponding interior particles to the virtual particles. These virtual particles are featured by:

- During the entire simulation, not only the X axis distance between the virtual particles (e.g., particle K in Fig. 1) which are closest to interior particles and the corresponding interior particles stays at d_0 but also the X axis distance between any two adjacent virtual particles on the same line also stays at d_0. And the Y axis coordinate of the virtual particles on the same line always equal to the Y axis coordinate of the corresponding interior particles

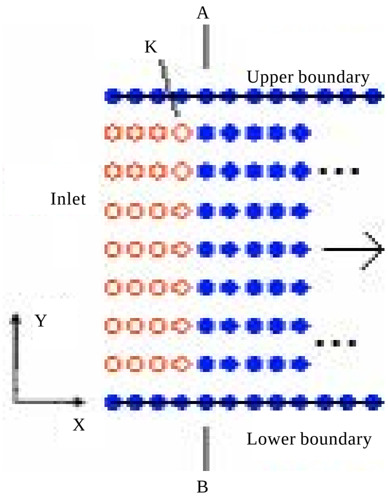

Fig. 1: Diagram of virtual following particles

- In the entire simulation process, the mass, velocity and smoothing length of the virtual particles are all the same with those of the corresponding interior particles. The calculation methods for the density and dynamic pressure of the virtual particles are the same as that of interior particles
- The virtual particles also participate in the interactions with other particles (including virtual particles and interior particles)

As these virtual particles move with the corresponding interior particles during simulation process, they are referred to as virtual following particles.

Similarly, the virtual following particles can also be set at the outlet boundaries to implement the laminar flow boundary conditions, however, they are on the right of the interior particles.

TEST CASES

This study verifies the validness of the proposed method by means of two test cases of incompressible flows (the liquid is water). One test case is the Poiseuille flow, the other is a two-dimensional flow between two inclined plates. The dynamic pressure is calculated by $p = c^2\rho$ (c is the artificial sound speed).

Poiseuille flow: In the model of Poiseuille flow (Morris *et al*., 1997), the liquid flows between stationary two parallel and infinite plates located at $y = 0$ and $y = L$, respectively.

In the Poiseuille flow, apart from the method of treating inlet and outlet by means of periodic boundary conditions, taking a section of the plates and treating the inlet and outlet boundaries with virtual following particles can also obtain the same simulation results. In the following case, under the condition when other conditions are the same, treat the outlet and inlet boundaries by means of the two methods, respectively, then compare whether the results obtained by the two methods are identical. This is one of the effective methods to verify the validness of the proposed method.

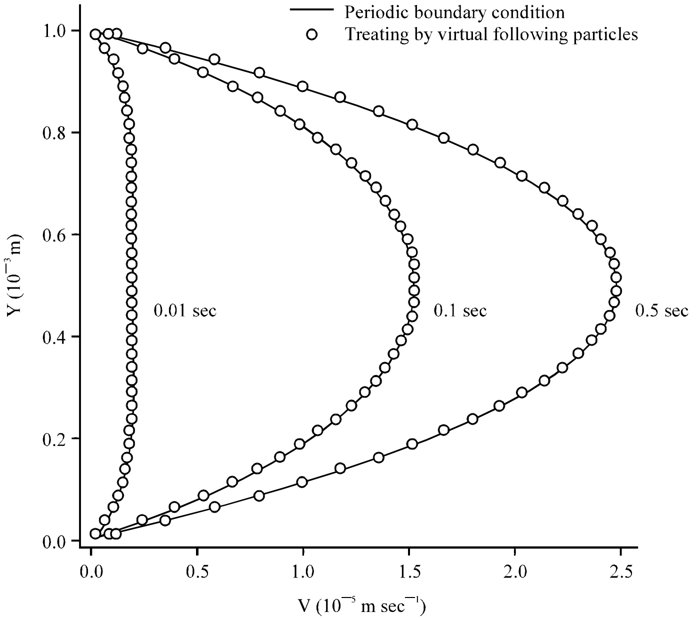

Fig. 2: Comparison of the horizontal velocities obtained by two methods

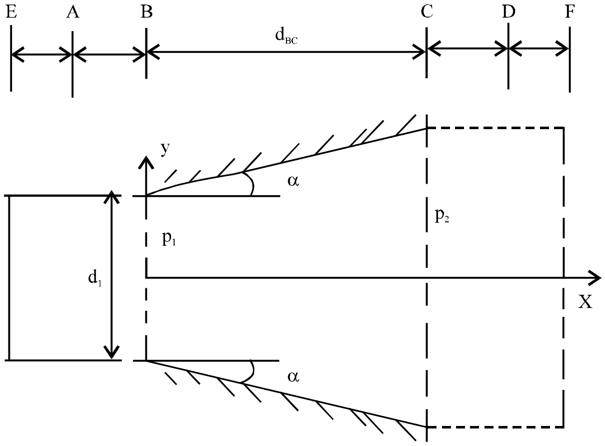

Fig. 3: Model of two-dimensional flow between two inclined plates

Figure 2 provides the results obtained by the two methods under the condition that other parameter conditions were the same. The body force of the liquid was $F = 2\times10^{-4}$ m sec^{-2}.

The diagram shows that the results from the proposed method were in good agreement with those from the method of periodic boundary condition. Here, the Reynolds number was Re = 0.025.

Two-dimensional flow between two inclined plates:

- **Calculation model:** As shown in Fig. 3, given $\alpha = 3.503°$ and give the boundaries are stationary (solid boundary), the differential pressure between cross section B and C is given as $\Delta p = p_1 - p_2$

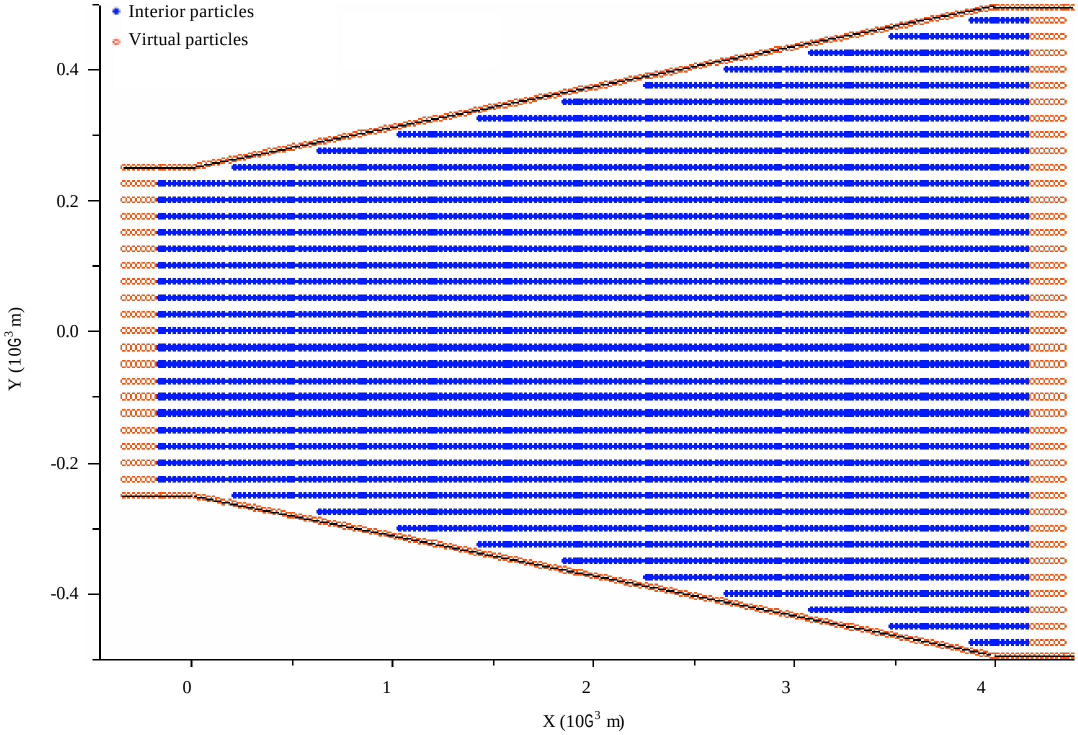

Fig. 4: Diagram of initial setting for particles

To conduct SPH simulation for the flow field in Fig. 3, there are two problems need to be solved. Firstly, since the sizes of the inlet and outlet are inconsistent, it's obvious that periodic boundary condition cannot be applied, then how to treat the inlet and outlet boundaries becomes the first problem. The second problem is, how to convert the differential pressure between the two ends of the flow field into the body force of the particles. The second problem can be solved by the method in literature (Liang *et al.*, 2012) and this study will focus on solving the first problem.

For the first problem, extension to both sides as shown in Fig. 3 can be considered, where particles are set in AB section and the body force of the particles are the same as that of the particles in cross section B. Particles are also set in CD section and the body force of the particles are the same as that of the particles in cross section C. Virtual following particles are set in EA and DF section.

- **Simulation parameters and particle setting**

  Simulation parameters: Differential pressure p = 1.21665968×10^{-3} N/m^2, d_{BC} = 4 mm, d_1 = 0.5 mm. The quintic spline function shall be selected as the kernel function, dt = 10^{-4} sec, h = d_0, d_0 = 2.5×10^{-5} m, artificial sound velocity c = 2.5×10^{-4} m sec^{-1}. Take AB = CD = EA = DF = $7d_0$. The initial velocity and initial dynamic pressure of all particles are zero. The initial particle arrangement is shown in Fig. 4. The mass and initial density of the virtual particles on the stationary boundary (solid wall) are the same as that of the interior particles and their positions stay unchanged during simulation, with the velocity of zero

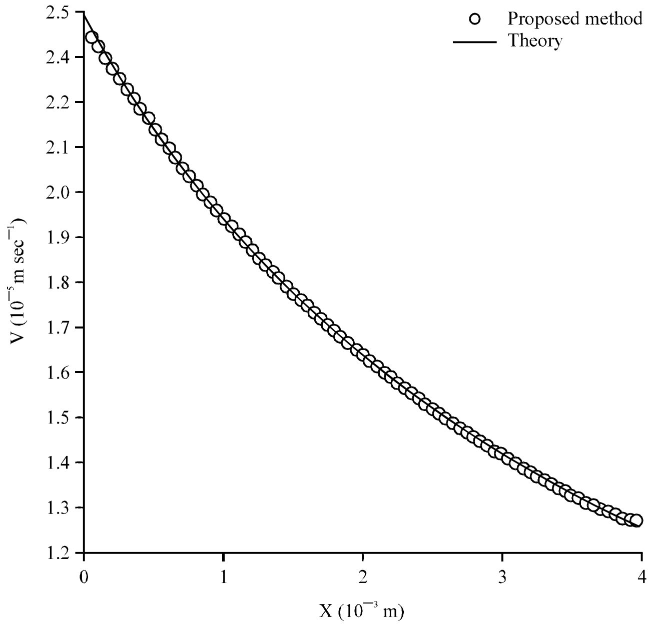

Fig. 5: Comparison of the horizontal velocities on X axis between the proposed method and the theoretical value

all the time, meanwhile, they apply short distance radiation repulsive force on the interior particles to prevent the particles from penetrating the solid boundary (Monaghan, 1994). To get higher precision, the Finite Particle Method (FPM) which can overcome the uneven setting of particles and the truncation of the boundary is selected (the second derivative of Taylor expansion is retained in this method) (Liu *et al.*, 2005)

- **Simulation results:** It reached a steady state after running for 4000 steps. Figure 5 shows the fluid's horizontal velocity at X axis in Fig. 4 when running for 4000 steps

For the flow field in Fig. 3, by referencing the derivation in literature (Lin *et al.*, 2005), the theoretical value of the fluid's horizontal velocity can be obtained as follows (Liang *et al.*, 2012):

$$u_x = -\frac{1}{\mu}\left[y^2 - \frac{(d_1 + 2x\tan\alpha)^2}{4}\right]\frac{(d_1)^2(d_1 + 2d_{BC}\tan\alpha)^2}{d_{BC}(d_1 + d_1 + 2d_{BC}\tan\alpha)(d_1 + 2x\tan\alpha)^3}\Delta p \quad (3)$$

As seen from Fig. 5, the numerical solution was in good agreement with the theoretical value. The Reynolds number here was about 0.01.

CONCLUSION

This study proposes a method of treating the inlet or outlet boundaries of laminar flow with virtual following particles under low Reynolds number. The Poiseuille flow and two-dimensional flow between two inclined plates were simulated, respectively with the proposed method and the results were in good agreement with theoretical values which verifies the validness of the proposed method. This boundary treating method can be applied to low Reynolds number flow fields with a laminar flow inlet or outlet.

ACKNOWLEDGMENT

This study is supported by the Department of Education of Guangxi Zhuang Autonomous Region of China under Grant No. 2013YB224.

REFERENCES

Liang, C.X., C.Y. Zhang, H.Y. Liu, Z.R. Qin and Q. Wang, 2012. Modeling low Reynolds number incompressible flows with curved boundaries using SPH. Int. J. Numer. Methods Fluids, 68: 1173-1188.

Lin, J.Z., X.D. Ruan and B.G. Chen, 2005. Fluid Mechanics. Tsinghua University Press, Beijing, China, pp: 263-270.

Liu, G.R. and M.B. Liu, 2003. Smoothed Particle Hydrodynamics: A Meshfree Practical Method. World Scientific Publishing, Singapore, pp: 114, 134-137, 147.

Liu, M.B., W.P. Xie and G.R. Liu, 2005. Modeling incompressible flows using a finite particle method. Applied Math. Modell., 29: 1252-1270.

Monaghan, J.J., 1994. Simulating free surface flows with SPH. J. Comput. Phys., 110: 399-406.

Morris, J.P., P.J. Fox and Y. Zhu, 1997. Modeling low Reynolds number incompressible flows using SPH. J. Comput. Phys., 136: 214-226.

Rafiee, A., M.T. Manzari and M. Hosseini, 2007. An incompressible SPH method for simulation of unsteady viscoelastic free-surface flows. Int. J. Non-Linear Mech., 42: 1210-1223.