

SPH no-slip BC implementation analysis at the continuous level

A. Souto-Iglesias
L.M. González

Naval Architecture Dept.(ETSIN)
Technical University of Madrid (UPM)
28040 Madrid, Spain
antonio.souto,leo.gonzalez@upm.es

A. Colagrossi
INSEAN

00128 Rome, Italy
CeSOS, NTNU,
Trondheim, Norway
a.colagrossi@insean.it

M. Antuono
INSEAN

00128 Rome, Italy.
matteoantuono@gmail.com

Abstract—The aim of this paper is to discuss whether there are structural problems in three of the most representative velocity mirroring techniques used to force no-slip boundary conditions (BC) in SPH for Newtonian incompressible flows. We refer specifically to fixed fluid particles, ghost particles, and Takeda *et al.* [1] boundary integrals. In Newtonian incompressible flows, the viscous related terms in the momentum conservation equation depend on the evaluation of the Laplacian of the velocity field. In order to analyze such techniques, the continuous version of the Laplacian approximation by Morris *et al.* [2] and Monaghan-Cleary-Gingold [3] viscous terms has been considered. It has been shown that there are intrinsic inaccuracies in the computation of the Laplacian close to the boundaries and the onset of singularities in such evaluation for some flows and mirroring techniques combinations. The impact of these deviations in the SPH simulation of viscous flows is not clear at this stage.

SYMBOLS

MVT	Morris <i>et al.</i> viscous term [2]
MCGVT	Monaghan-Cleary-Gingold [3] viscous term
BC	Boundary condition(s)
FFP	Fixed fluid particles
GP	Ghost particles
ASM	Anti-symmetry model for velocity mirroring
UOM	Zero velocity mirroring model

I. INTRODUCTION

The modeling of no-slip BC plays an important role in the simulation of many important physical phenomena like boundary layers, separation, transition flows, etc., and in the computation of important magnitudes in Engineering like the viscous drag force. Our aim with this paper is to discuss whether a set of very representative mirroring techniques of the velocity field used to force no-slip BC for Newtonian incompressible flows have essential problems in doing so, at the continuous level of the SPH formulation. In Newtonian incompressible flows, the viscous related terms in the momentum conservation equation depend on the evaluation of the Laplacian of the velocity field, which presents difficulties in general, and in particular, close to the boundaries. Therefore, the analysis should focus not in the value of the velocity at the boundary but in the value of the Laplacian. Although

some general results are outlined, the analysis presented here is treats mainly linear and quadratic velocity fields.

In a previous paper [4], we had already started the study of this problem by analyzing the accuracy of the continuous version of Monaghan-Cleary-Gingold viscous term [3] for the evaluation of the Laplacian of quadratic velocity fields. The ghost particle (GP) technique had then been considered. The analysis had a limited scope but some incongruities in the evaluation of the Laplacian close to the boundaries were shown. In the present paper, such analysis is extended by using as well the general formulation of Español and Revenga [5] for the computation of the Laplacian. The SPH discretization of such term leads to the well know Morris *et al.* viscosity term [2]. On top of this extension, fixed fluid particles as well as the evaluation of the boundary effects with Takeda *et al.* [1] integrals have been considered.

The paper is organized as follows: first, the field equations and the SPH formalism considered is presented and discussed. Second, the selected implementations of no-slip BC techniques are introduced. Third, two 2D given velocity fields are considered in order to check the performance of the viscous terms at the continuous level close to the boundaries. Finally the evolution of a plane Couette and Poiseuille flows is studied in order to assess the validity of the conclusions obtained from the previous analysis.

II. GOVERNING EQUATIONS

A. Field equations

The aim of this paper is to deal with Newtonian incompressible flows. The incompressible Navier-Stokes equations in Lagrangian formalism are hence taken as the field equations:

$$\frac{D\mathbf{r}}{Dt} = \mathbf{u}, \quad \nabla \cdot \mathbf{u} = 0, \quad \frac{D\mathbf{u}}{Dt} = \mathbf{g} + \frac{\nabla \cdot \mathbb{T}}{\rho}, \quad (1)$$

In these equations, ρ is the fluid density and \mathbf{g} is a generic external volumetric force field. The flow velocity, \mathbf{u} , is defined as the material derivative of a fluid particle position \mathbf{r} . \mathbb{T} is the stress tensor of a Newtonian incompressible fluid:

$$\mathbb{T} = -p\mathbf{I} + 2\mu\mathbb{D}, \quad (2)$$

in which p is the pressure, \mathbb{D} is the rate of deformation tensor - $\mathbb{D} = (\nabla\mathbf{u} + \nabla\mathbf{u}^T)/2$ - and μ is the dynamic viscosity. With this notation, the divergence of the stress tensor, \mathbb{T} is computed as:

$$\nabla \cdot \mathbb{T} = -\nabla p + \mu \nabla^2 \mathbf{u}. \quad (3)$$

The computation of the viscous effects is therefore directly related with the computation of the Laplacian of the velocity field, as aforementioned.

B. Boundary conditions (BC)

In order to close the system of equations (1), it is necessary to specify the BC. The fluid domain is denoted as Ω and its boundary is denoted as $\partial\Omega$. Only solid boundaries (no free surface), on which a no-slip BC is imposed, are of interest for the present study. Such condition is expressed in the equation (4) in which $\mathbf{V}_{\partial\Omega}$ designates the boundary velocity.

$$\mathbf{u} = \mathbf{V}_{\partial\Omega} \quad \forall \mathbf{r} \in \partial\Omega \quad (4)$$

III. THE CONTINUOUS SPH MODEL

A. General

In order to introduce the notation, a brief description of prominent aspects of the SPH formulation is presented (see [6] for a comprehensive review). The SPH continuous model is based on the filtering (*smoothing*) of any generic flow field f with a convolution integral over the fluid domain Ω

$$\langle f \rangle(\mathbf{r}) = \int_{\Omega} f(\mathbf{r}') W(\mathbf{r}' - \mathbf{r}; h) dV' \quad (5)$$

$W(\mathbf{r}' - \mathbf{r}; h)$ is a weight function, (*kernel* in the SPH lexicology), which in practical applications has a compact support $\Omega(\mathbf{r})$. h (usually referred to as the *smoothing length*) is a characteristic length of $\Omega(\mathbf{r})$ (figure 1). Even for kernels without compact support, like the Gaussian one used in some of the analysis of the present paper, the value of h indicates to what extent the closer points to \mathbf{r} matter in the integral 5. The smaller h , the larger weight of nearest points. h is considered strictly constant in this paper in the sense that its spatial and time derivatives are therefore identically zero. The weight function $W(\mathbf{r}' - \mathbf{r}, h)$ is positive, radial centered in \mathbf{r} and decreases monotonously with the distance $s = |\mathbf{r} - \mathbf{r}'|$. The kernel in this study is supposed to be isotropic, which implies that depends only on the distance s . In the limit as the smoothing length h goes to zero, the original field of the convolution integral (5) should be recovered. In order to be so, the kernel W must integrate to one [7]. Such a property is not satisfied when the kernel domain is not completely immersed inside the fluid domain [7], which is a quite common configuration for those particles within a specific distance from the domain boundary $\partial\Omega$ (figure 1).

If the smoothing procedure is applied to the differential operators of the governing equations (1), shortening the notation $\langle f \rangle(\mathbf{r})$ by $\langle f \rangle$, the SPH continuous formulation of the Navier-Stokes equations is obtained. In this work, we focus on the momentum equation.

$$\frac{D\mathbf{u}}{Dt} = \mathbf{g} - \frac{\langle \nabla p \rangle}{\rho} + \frac{\mu \langle \nabla^2 \mathbf{u} \rangle}{\rho} \quad (6)$$

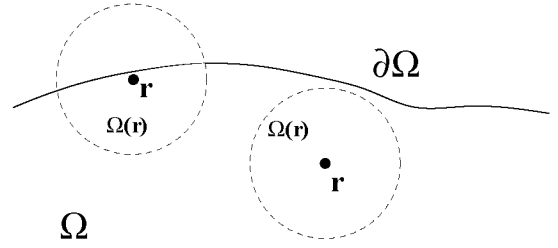


Fig. 1. Configurations of the kernel support $\Omega(\mathbf{r})$ with respect to the fluid domain boundary.

B. Continuous SPH viscous term

Español and Revenga [5] deduced a general expression for second derivatives that can be particularized for the Laplacian of the velocity field as:

$$\langle \nabla^2 \mathbf{u} \rangle(\mathbf{r}) = 2 \int_{\Omega} \frac{(\mathbf{r}' - \mathbf{r}) \nabla W(\mathbf{r}' - \mathbf{r}, h)}{|\mathbf{r}' - \mathbf{r}|^2} [\mathbf{u}(\mathbf{r}') - \mathbf{u}(\mathbf{r})] dV' \quad (7)$$

The derivative $\nabla W(\mathbf{r}' - \mathbf{r}, h)$ refers to the variable \mathbf{r} . It takes the compact form (8) due to the kernel isotropy.

$$\nabla W(\mathbf{r}' - \mathbf{r}, h) = -\frac{\mathbf{r}' - \mathbf{r}}{|\mathbf{r}' - \mathbf{r}|} \frac{dW}{ds} \quad (8)$$

If $\mathbf{u}(\mathbf{r})$ and $\mathbf{u}(\mathbf{r}')$ are respectively shortened as \mathbf{u} and \mathbf{u}' , the integral (7) is abbreviated as:

$$\langle \nabla^2 \mathbf{u} \rangle(\mathbf{r}) = 2 \int_{\Omega} \frac{\mathbf{u}' - \mathbf{u}}{|\mathbf{r}' - \mathbf{r}|} \frac{dW}{ds} dV' \quad (9)$$

When seen from the discrete point of view, this viscous term gives contributions to the Laplacian parallel to the velocity differences between interacting particles. The straightforward discretization of this term is the Morris et al. [2] one and we will refer to it hereafter as MVT.

There is another well known possibility that derives from the artificial viscosity term [6], [8], [9] in which the contributions to the Laplacian are radial. Hu and Adams [10] demonstrated that when the fluid is assumed to be incompressible, this term can be obtained from Español and Revenga [5] formula. The continuous SPH formulation of this second expression takes the following form.

$$\frac{\langle \nabla^2 \mathbf{u} \rangle(\mathbf{r})}{2(d+2)} = \int_{\Omega} \frac{[\mathbf{u}(\mathbf{r}') - \mathbf{u}(\mathbf{r})] \cdot (\mathbf{r}' - \mathbf{r})}{|\mathbf{r}' - \mathbf{r}|^2} \nabla W(\mathbf{r}' - \mathbf{r}, h) dV' \quad (10)$$

where d is number of dimensions. With the notation of formula (9), we get to the following expression in 2D:

$$\langle \nabla^2 \mathbf{u} \rangle(\mathbf{r}) = -8 \int_{\Omega} \frac{(\mathbf{u}' - \mathbf{u}) \cdot (\mathbf{r}' - \mathbf{r})}{|\mathbf{r}' - \mathbf{r}|^3} (\mathbf{r}' - \mathbf{r}) \frac{dW}{ds} dV', \quad (11)$$

Cleary and Monaghan [9], [11] were the first to use this term as a shear viscosity. The term will be referred to hereinafter as MCGVT, using the authors's initials and the fact it was devised in Melbourne, home of the old beautiful MCG stadium.

In regards to the order of the smoothed approximations (5), (7) and (11), we refer the reader to references [5], [6], [10] respectively.

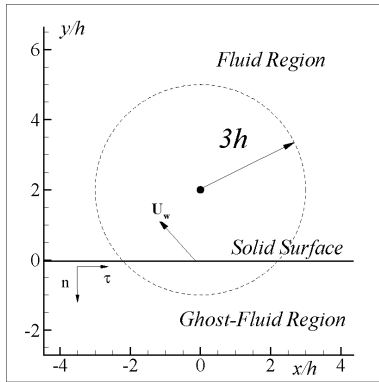


Fig. 2. Sketch of the interpolation support near a solid boundary

IV. NO SLIP BC IMPLEMENTATIONS

A. General

Although there have been recently some new more sophisticated formulations [12], [13], the most representative models for no-slip solid boundary conditions in SPH are: Rows of fixed (zero velocity) fluid particles (FFP) [3], ghost particles (GP) [14], [15] and the Takeda *et al.* [1] imaginary particles. These techniques are discussed in the next sections. They are based on defining a virtual velocity field across the boundary, which later enters into the SPH summations. This may affect not only the no-slip BC but others like the Neumann BC on the pressure or even the normal velocity one. The analysis of these issues is left for a future study.

B. Fixed fluid particles (FFP)

No specific treatment of no-slip is necessary when modeling solid boundaries with fully interacting fluid particles that are assigned the solid boundary velocity. This technique is straightforward to implement and has been used for instance by Monaghan [3] for modeling a transient Couette flow. The fixed fluid particle (FFP) is, from the continuous perspective, equivalent to the GP technique taking the velocity of the GP as zero (U0M). Therefore, its study will be contained in the one dedicated to the GP.

C. Ghost particles (GP)

One of the most widespread method to treat the solid BC is based on the use of the so-called *ghost particles* (e.g. [14], [15]). In the GP technique a set of fictitious particles mirror, one by one, the actual fluid particle properties with respect to the solid boundary (figure 2). The normal velocity of the GP is obtained by linearly extrapolating the normal velocity of the original fluid particle in such a way that the normal velocity at the wall will be the normal velocity of the wall. This is formalized in the equation (12), in which \mathbf{n} is the unitary normal of the solid boundary, pointing outwards of the fluid domain Ω , \mathbf{U}_w is the solid boundary velocity, and \mathbf{u}^G and \mathbf{u}^F are respectively the ghost and fluid particle velocities.

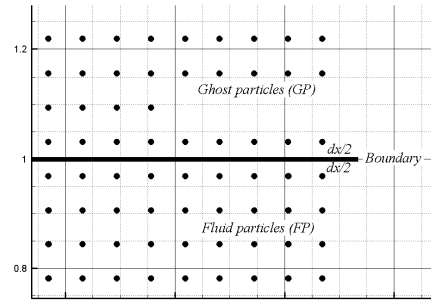


Fig. 3. GP arrangement (dx is the typical particle spacing)

$$\mathbf{u}^G \cdot \mathbf{n} = (2\mathbf{U}_w - \mathbf{u}^F) \cdot \mathbf{n}, \quad (12)$$

The difficulties arise from the implementation of the tangential no-slip BC, physically related with the effects of viscosity at the boundaries. A first possibility, consistent with the GP normal velocity definition, is an anti-symmetric mirroring of the tangential flow velocity with respect to the solid boundary. This approach will be referred to as the ASM procedure and has been formalized in equation (13) in which $\boldsymbol{\tau}$ is the tangential -to the solid boundary- unit vector.

$$\mathbf{u}^G \cdot \boldsymbol{\tau} = (2\mathbf{U}_w - \mathbf{u}^F) \cdot \boldsymbol{\tau}, \quad (13)$$

In the second procedure, it is assumed that the GP have the same tangential velocity as the solid boundary. This procedure will be designated as U0M and it is expressed mathematically with the equation (14).

$$\mathbf{u}^G \cdot \boldsymbol{\tau} = \mathbf{U}_w \cdot \boldsymbol{\tau}, \quad (14)$$

The practical consequence of the mirroring procedure is that a new set of GP is created at every time step, which enter into the summations as if they were fluid particles. The initial arrangement of the particles in an SPH simulation, which will be mimicked to perform the numerical integrals of the Laplacian, is presented in figure 3.

With the ASM procedure, it is possible to demonstrate that the computed Laplacian on the solid boundary is null both for MCGVT and MVT. This is a consequence of the fact that the *global* velocity field (that is, the velocity field of the fluid particles on one side of the boundary and of the GP on the outwards side) is *odd* with respect to the solid boundary. Since the the rest of the factors in the integrand, are as whole, an *even* factor themselves, the smoothing integral returns a null integral. A rigorous demonstration of this matter is left for future studies.

D. Takeda *et al.* [1] model

No-slip BC were treated by Takeda *et al.* [1] by analytically estimating the influence of imaginary particles at the other side of the boundary, for which an anti-symmetric velocity field has been imposed. Due to this fact, this model could be thought, from the continuous point of view, to be equivalent to the GP technique with anti-symmetric modeling (GP-ASM).

Nonetheless, this is not the case; in the GP-ASM model, the fluid particles, all “see” the same GP at the other side of the boundary. In the Takeda *et al.* [1] model, the imaginary particles velocity field is different for each fluid particle, obtained by mirroring the fluid particle velocity vector.

V. RESULTS

A. General

In order to check to what extent the selected implementations of the no-slip BC are able to correctly reproduce the Laplacian of a specific problem, we have selected a set of 2D tests cases. They comprise on one side two given velocity fields and two realistic fluid dynamics problems. In the given velocity cases (a linear and a quadratic field), the continuous version of Laplacian operators for the MVT and MCGVT has been evaluated near a solid boundary with an “exact” numerical integration (using a Gaussian kernel) and assessing the dependence on h . The exact values of the Laplacian for those given flows should be recovered as h goes to zero. Since the results in the computation of the Laplacian for these imposed velocity fields show some inaccuracies close to the solid boundaries, it makes sense to check to what extent those inaccuracies may affect realistic flows. To do so, two well known solutions of the Navier Stokes equation, plane Couette and Poiseuille flows have been considered, and have been simulated with a standard SPH code. They have a steady state solution which is either linear or quadratic and this makes them good candidates for the tests. The exact steady state solution has been imposed as initial condition to see to what extent this condition is compatible with the SPH modeling of viscous terms and solid BC.

In the SPH simulations of the present paper, in order to make the simulation coherent with the Navier-Stokes exact solution hypothesis, the pressure terms have been removed from the momentum equation. Furthermore, vertical velocities are made zero. Since the results of the given fields indicate that the viscous terms MVT and MCGVT perform similarly at the continuous level, and for the sake of brevity, MCGVT results have not been included in this part. A $3h$ support renormalized Gaussian kernel [16] has been used. The particles initial configuration is a regular lattice (figure 3).

B. Linear field

1) *General*: A linear horizontal velocity field $u(y) = y$ is considered. The exact value of the velocity Laplacian u_{yy} is therefore zero. The solid boundary is supposed to be at $y = 0$.

2) *GP*: As anticipated in section IV-C, the ASM SPH approximation of the Laplacian should be exactly zero for a linear velocity field. This has been actually the case in the numerical computation. The results are quite different for UOM model. The Laplacian is not zero and therefore incorrect, as can be appreciated in figure 4, where the evaluation of u_{yy} near the solid boundary ($y = 0$) for the linear flow is presented. There is actually a singularity with $1/h$ in the numerical evaluation of this integral as can be appreciated from those

figures, by noticing that as h halves, the value of the Laplacian, as y goes to zero, doubles. This conclusion is independent of the viscosity model (MVT or MCGVT).

3) *Takeda et al. [1] model*: The solution is exact for the linear field since in this case, Takeda *et al.* [1] model is equivalent to the GP-ASM model.

C. Quadratic field

1) *General*: A quadratic field of the type $u(y) = y^2/2$, near a flat solid boundary ($y = 0$), is considered. The exact value of the velocity Laplacian u_{yy} is identically equal to 1.

2) *GP*: The ASM model provides, coherently with what was anticipated in section IV-C, a null (incorrect) value of the Laplacian at the boundary, but converges to the exact one as the interpolation point moves away from the boundary (figure 5). The results for UOM technique are incorrect at the boundary (0.5 instead of 1) but no singularity is found, contrary to what happens with the linear field (see section V-B2). Preliminary tentative analysis indicate that the singularity onsets only for linear field. Also in the UOM technique, as the evaluation point moves away from the boundary, the result converges to the exact one. These conclusions are independent of the viscous term model (MVT or MCGVT) as can be clearly appreciated in figures 5, 6.

3) *Takeda et al. [1] model*: The solution is not exact for the quadratic velocity field (figure 7). The value of the Laplacian is actually halved close to the solid boundary, but at least it is not identically zero, as with the GP ASM model. It is relevant that this is the case, since the Takeda *et al.* [1] model is based as well on a kind of anti-symmetrization of the velocity field. MVT and MCGVT perform similarly.

D. Plane Couette flow

1) *General*: The evolution from rest of a Couette flow was treated by Monaghan [3] finding satisfactory results for the velocity field using FFP. We depart from the steady state solution and assess whether this linear field is accepted by the SPH simulation. The energy is transferred through the viscous stresses from the wall onto the fluid mass till a point where the viscous dissipation equals the energy transfer from the walls. The flow is considered x -periodic, x being the longitudinal channel coordinate. The transversal coordinate is noted y . The elementary fluid domain is taken as $D = (-L/2, L/2) \times [0, B]$, with $L = 2$, $B = 1$. The motion is induced by the constant velocity U_w displacement of the top wall. The exact stationary laminar solution does not depend on x nor on the viscosity; the exact velocity field has zero vertical component and linear horizontal one.

$$u(y) = U_w \cdot y \quad (15)$$

A laminar regime with Reynolds number 32 (like in reference [3]) is considered in the simulations. The Reynolds number is based on the top boundary velocity, U_w and the channel width B , all equal to one. Density ρ is also taken as one. The simulation is stopped when, after n_c consecutive time steps (n_c of the order of 10), the condition expressed in equation (16),

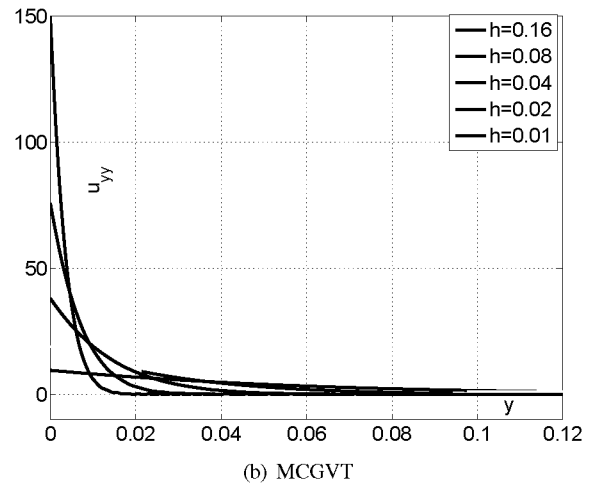
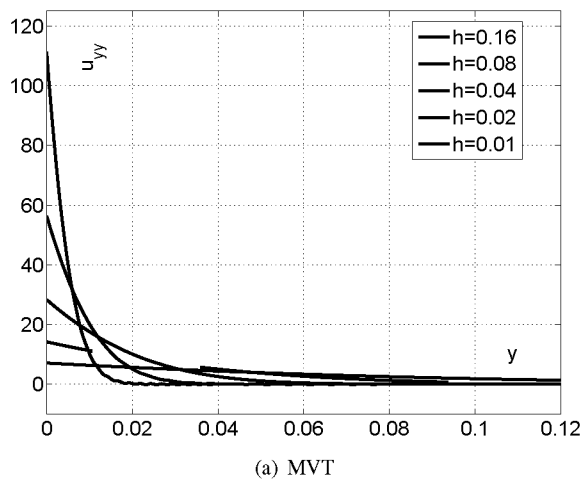


Fig. 4. Linear field: GP UOM Laplacian. h dependence.

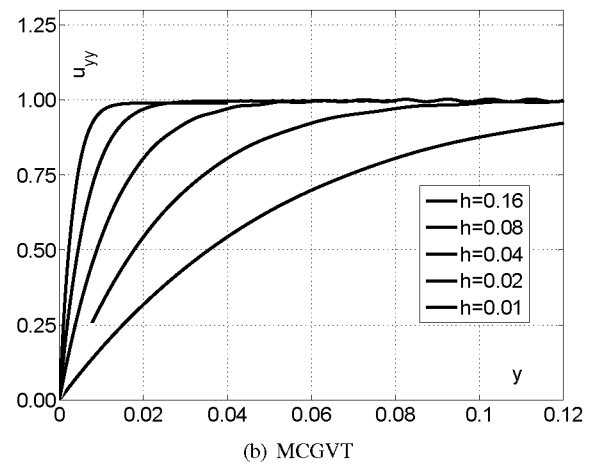
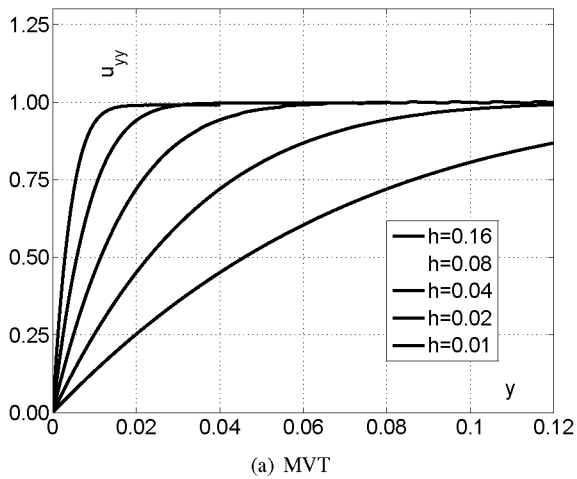


Fig. 5. Quadratic field: GP ASM Laplacian. h dependence.

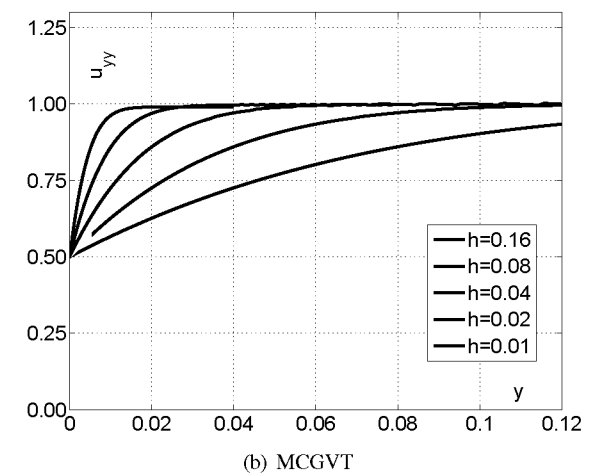
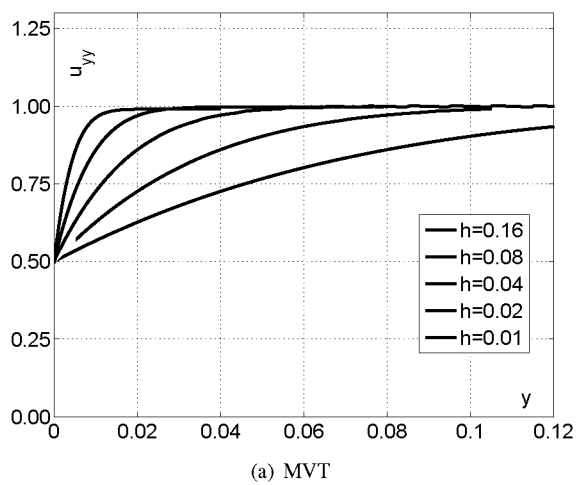


Fig. 6. Quadratic field: GP UOM Laplacian. h dependence.

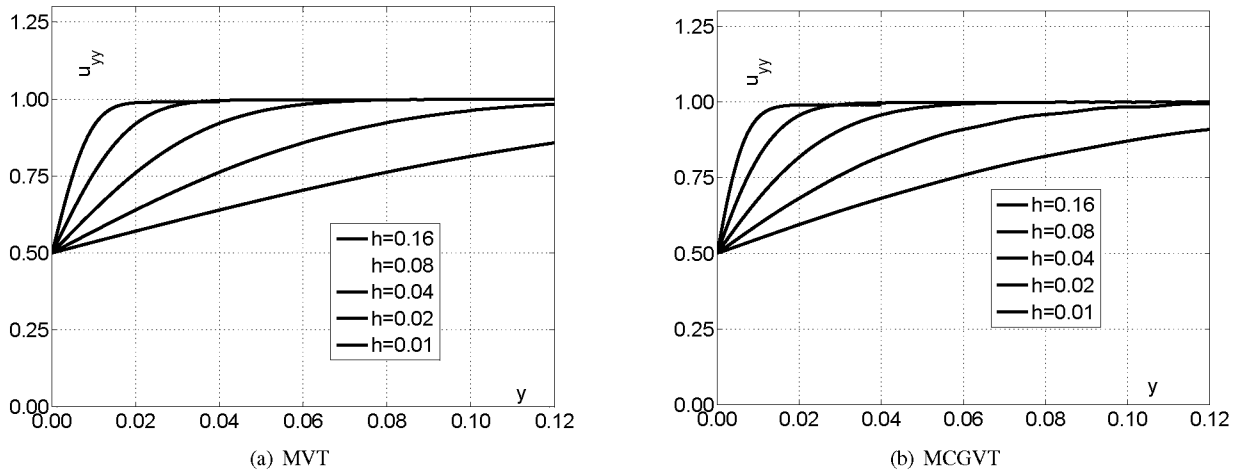


Fig. 7. Quadratic field: Takeda *et al.* [1] Laplacian. h dependence.

relative to the kinetic energy E_k , is fulfilled, with E_0 being its exact value.

$$\frac{|E_k^{n+n_c} - E_k^n|}{E_0} < 0.00001 \quad (16)$$

A convergence analysis, both as h and dx/h simultaneously tend to zero is provided in order to establish the validity of the conclusions at the continuous level.

2) *GP*: The ASM model accepts the exact solution exactly, as expected, and no perturbation occurs in the velocity field. Coming to the UOM, the exact solution is not accepted by the model. The walls do not transfer enough energy to the fluid which in turn dissipates more energy from viscosity. As a consequence, the total kinetic energy of the fluid diminishes, as can be seen in figure 8(a), corresponding to MVT. From that curve it seems that as the smoothing length tends to zero, the deviation of kinetic energy from the exact one tends as well to zero. Nonetheless, if we focus on the very first section of figure 8(a), by zooming on it (fig 8(b)), it can be seen that the slope of all the curves is constant and finite. This could mean that full convergence to the exact solution might not take place, regardless of the resolution.

3) *Takeda et al. [1] model*: The solution is exact for this problem, coherently with the results of this technique for the given linear field, discussed in section V-B3.

E. Plane Poiseuille flow

1) *General*: As in the Couette test case (section V-D), the flow is considered x -periodic in the velocity field. The elementary fluid domain, the notation, and the SPH implementation are the same like in that case. Only the stopping criterium is modified, changing the right hand side of the inequality (16) to $1e-6$, due to the evolution of the flow being slower. The motion is induced by an uniform pressure x -gradient $\partial P/\partial x$. The velocity profile of the exact stationary solution is presented in equation (17).

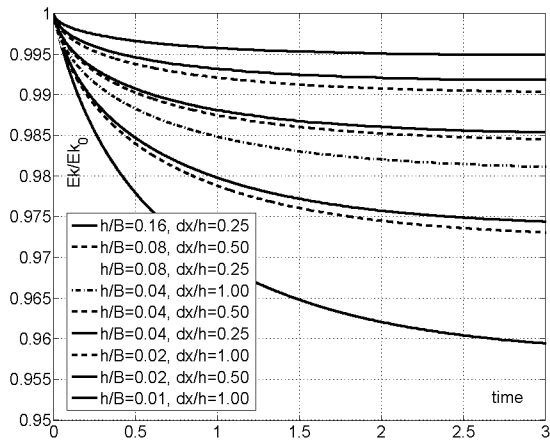
$$u(y) = \frac{\partial P}{\partial x} \frac{y(y-1)}{2\mu} \quad (17)$$

A laminar regime of $Re = 10$ has been chosen for these simulations. The Reynolds number Re is based in this case on the maximum velocity of the steady-state solution U_0 and the channel width. The pressure gradient has been taken as -1 and the fluid density has been taken as $+1$.

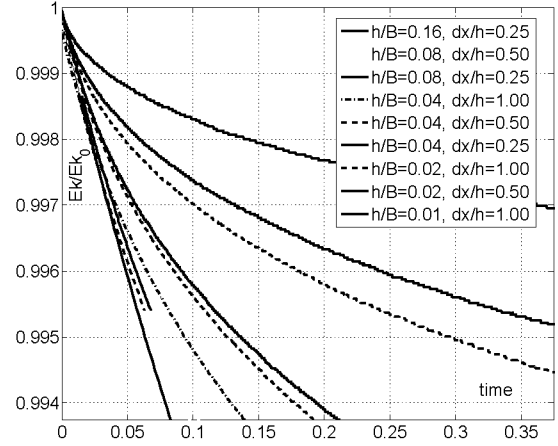
2) *GP*: Similarly to section V-D2 a convergence analysis using as indicator the kinetic energy is the main source of evidence for the assessment of the performance of the no-slip implementation models. Again, the initial condition is the steady state solution and we seek to assess how well this exact solution is accepted by the SPH model in regards to the accuracy of the no-slip implementation models. Results for MVT are presented in figures 9-10. In both ASM and UOM mirroring models, the deviation of the kinetic energy with respect to the exact one seems to converge to zero, as h goes to zero. Convergence is slow both for ASM and UOM models, though faster for the ASM model. The deviations of ASM simulations from the exact solution are smaller (around 10 times) than those of the UOM simulations, as can be appreciated from a comparative look at figures 9(a) and 10(a). Furthermore, the slopes of all the curves seem to tend to zero (figures 9(b) and 10(b)), contrary to what happened for the UOM in the plane Couette case.

In regards to the velocity field, a comparison with the exact solution for both mirroring models can be found in figure 11, computed with $h = 0.02$ and $dx/h = 0.5$. With these values, there are 100 particles in the span of the channel and the number of neighbors per particle is around 110. The errors are negligible for the ASM model (smaller deviation in kinetic energy from the exact solution as aforementioned). The errors are small but larger for the UOM, accordingly with a larger deviation in kinetic energy from the exact solution.

3) *Takeda et al. [1] model*: For the sake of brevity, and due to preliminar results indicating that the performance of this model may be qualitative and quantitatively similar to ASM model, an in-depth research on Takeda *et al.* [1] model performance for plane Poiseuille flow is left for future studies.

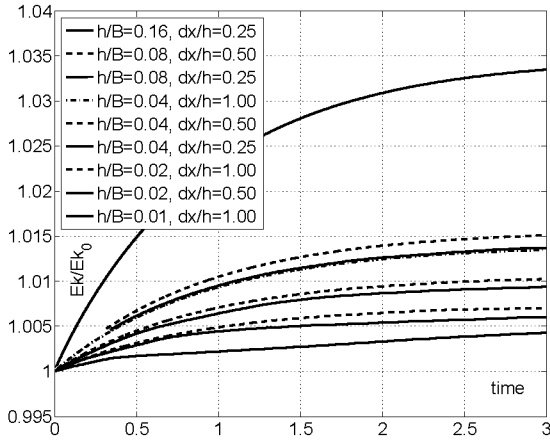


(a)

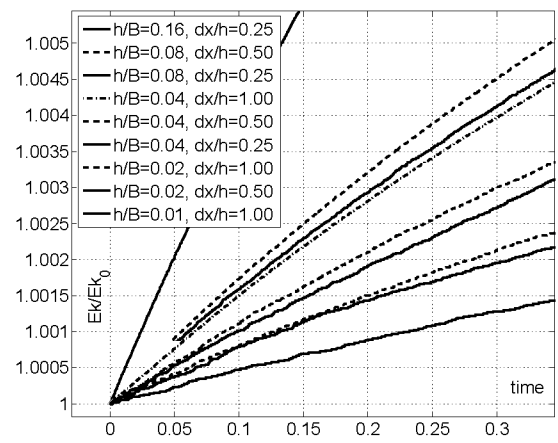


(b) zoom

Fig. 8. Couette flow: Kinetic energy, GP-U0M, MVT

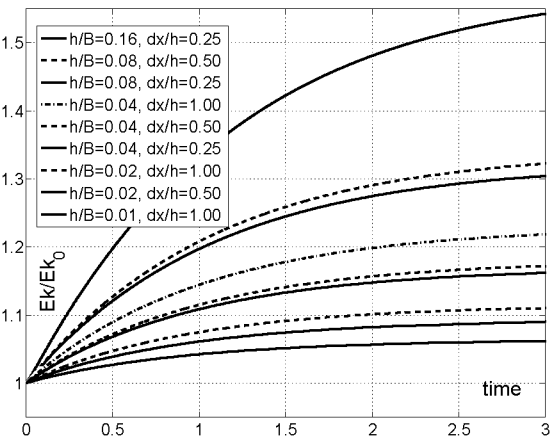


(a)

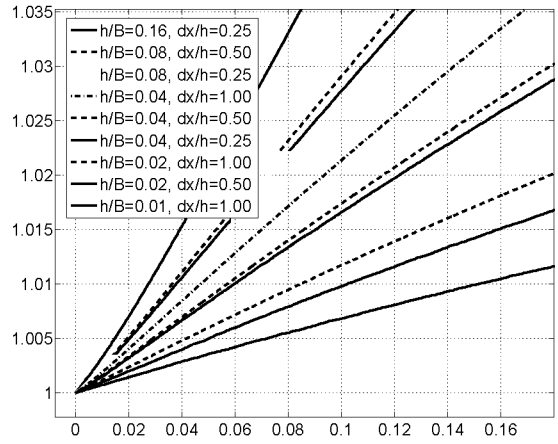


(b) zoom

Fig. 9. Poiseuille flow: Kinetic energy, MVT, GP-ASM



(a)



(b) zoom

Fig. 10. Poiseuille flow: Kinetic energy, MVT, GP-U0M

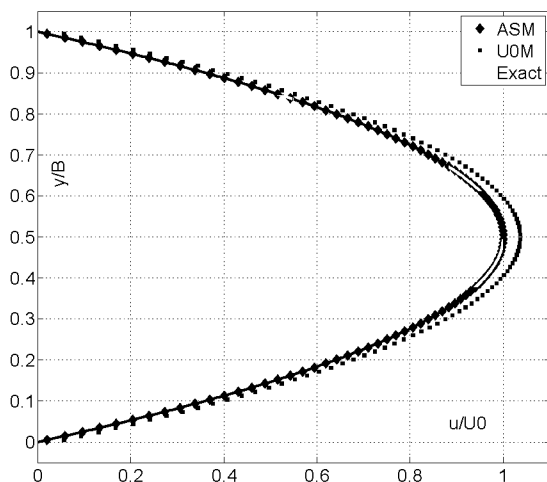


Fig. 11. Poiseuille flow: velocity, MVT, $h/B = 0.02$, $dx/h = 0.5$

VI. CONCLUSIONS AND FUTURE WORK

The aim of this paper has been to discuss whether there are some fundamental problems in the the most representative mirroring techniques of the velocity field used to force no-slip boundary conditions in SPH for Newtonian incompressible flows. We have specifically treated fixed fluid particles, Takeda *et al.* [1] boundary integrals and more specifically, ghost particles. Two equally representative viscous terms in its continuous integral form, namely Morris *et al.* viscosity term [2], and Monaghan-Cleary-Gingold [3] have been studied. No relevant difference has been found between them at the continuous level but it remains as future work the implications of our findings in their performance at the discrete level.

Basically linear and quadratic velocity flows have been considered. Though some hints are provided, extending the results to more general velocity fields is left for future studies. For the linear flows, it has been shown that both the ghost particle anti-symmetrization technique and the Takeda *et al.* [1] provide exact results, while for the zero velocity model, a singularity with the inverse of the smoothing length in the viscous term arises.

In regards to the quadratic velocity field, it has been shown that the ghost particle anti-symmetrization technique leads to an identically null (incorrect) value of the viscous terms at the boundary. A generalization of this result to any velocity field for the anti-symmetrization technique is left for future studies. Though quite similar in essence, the Takeda *et al.* [1] technique performs better in this case, providing a result which is not correct but which is not either zero at the boundary. The zero velocity model gives in this case an inaccurate result at the boundary but no singularity arises, as for the linear field.

We have tried to assess the influence of these findings in realistic fluid dynamics problems with a plane Couette and Poiseuille flows. The kinetic energy has been analyzed. Though some differences have been found that deserve further study, it seems that with that integral magnitude (kinetic

energy), the singular behavior is not observed, and that convergence to the exact solution seems feasible for all the models under study, though further work has yet to be done in order to clearly confirm such statement. It is left also for future studies the relevance of this study for practical SPH implementations with limited number of neighbors and disordered particles configurations

ACKNOWLEDGMENTS

The research leading to these results has received funding from the European Community's Seventh Framework Programme (FP7/2007-2013) under grant agreement n225967 "NextMuSE". This work was also partially supported by the Centre for Ships and Ocean Structures (CeSOS), NTNU, Trondheim, within the "Violent Water-Vessel Interactions and Related Structural Load" project.

REFERENCES

- [1] H. Takeda, S. M. Miyama, and M. Sekiya, "Numerical simulation of viscous flow by Smoothed Particle Hydrodynamics," *Progress of Theoretical Physics*, vol. 92, no. 5, pp. 939–960, 1994.
- [2] J. P. Morris, P. J. Fox, and Y. Zhu, "Modeling low Reynolds number incompressible flows using SPH," *Journal of Computational Physics*, vol. 136, pp. 214–226, 1997.
- [3] J. J. Monaghan, "Smoothed Particle Hydrodynamics simulations of shear flow," *Monthly Notices of the Royal Astronomical Society*, vol. 365, pp. 199–213, 2005.
- [4] A. Colagrossi, L. Delorme, G. Colicchio, A. Souto-Iglesias, and J. L. Cercós-Pita, "Reynolds number and shallow depth sloshing," in *3rd ERCOFTAC SPHERIC workshop*, June, 4-6 2008, pp. 221–228.
- [5] P. Español and M. Revenga, "Smoothed Dissipative Particle Dynamics," *Physical Review E*, vol. 67, 2003.
- [6] J. J. Monaghan, "Smoothed particle hydrodynamics," *Reports on Progress in Physics*, vol. 68, pp. 1703–1759, 2005.
- [7] A. Colagrossi, M. Antuono, and D. L. Touzé, "Theoretical considerations on the free-surface role in the Smoothed-particle-hydrodynamics model," *Physical Review E*, vol. 79, no. 5, p. 056701, 2009.
- [8] J. J. Monaghan and R. A. Gingold, "Shock simulation by the particle method SPH," *Journal of Computational Physics*, vol. 52, no. 2, pp. 374–389, 1983.
- [9] P. W. Cleary, "Modelling confined multi-material heat and mass flows using SPH," *Applied Mathematical Modelling*, vol. 22, no. 12, pp. 981–993, 1998.
- [10] X. Hu and N. A. Adams, "Angular-momentum conservative Smoothed Particle Hydrodynamics for incompressible viscous flows," *Phys. of Fl.*, vol. 18, pp. 702–706, 2006.
- [11] P. W. Cleary and J. J. Monaghan, "Boundary interactions and transition to turbulence for standard CFD problems using SPH," in *Proc. 6th International Computational Techniques and Applications Conference*, Canberra, 1993, pp. 157–165.
- [12] F. Bierbrauer, P. Bollada, and T. Phillips, "A consistent reflected image particle approach to the treatment of boundary conditions in smoothed particle hydrodynamics," *Computer Methods in Applied Mechanics and Engineering*, vol. 198, no. 41-44, pp. 3400–3410, 2009.
- [13] M. Basa, N. J. Quinlan, and M. Lastiwka, "Robustness and accuracy of SPH formulations for viscous flow," *International Journal for Numerical Methods in Fluids*, vol. 60, no. 10, pp. 1127–1148, 2009.
- [14] A. Colagrossi and M. Landrini, "Numerical simulation of interfacial flows by smoothed particle hydrodynamics," *J. Comp. Phys.*, vol. 191, pp. 448–475, 2003.
- [15] G. Liu and M. Liu, *Smoothed particle hydrodynamics. A meshfree particle method*. World Scientific Publishing Co. Pte. Ltd., 2003.
- [16] D. Molteni and A. Colagrossi, "A simple procedure to improve the pressure evaluation in hydrodynamic context using the SPH," *Computer Physics Communications*, vol. 180, pp. 861–872, 2009.