Investigation of wall functions in SPH for turbulent flow modelling

Aaron English and Renato Vacondio
Dept. of Engineering and Architecture
University of Parma
Parma, Italy
aaronrobert.english@unipr.it

ABSTRACT

Smoothed Particle Hydrodynamics (SPH) has shown promise in modeling turbulent flows, crucial for engineering applications, but faces challenges in accurately resolving near-wall turbulence without high computational costs. Existing SPH approaches, rely on high resolution near boundaries, which becomes infeasible for high Reynolds number flows. To address this, we extend the modified Dynamic Boundary Conditions (mDBC) method available in the open-source DualSPHysics code to compute wall shear stresses using a first-order consistent SPH spatial interpolation, enhancing near-wall accuracy. Furthermore, wall functions are introduced within the SPH framework, to model turbulence near boundaries while significantly reducing computational demands. Developed using the open-source DualSPHysics code, this approach represents a first attempt towards efficient and accurate wall modelling for turbulent flows. Preliminary analysis shows encouraging results for 3D channel flows and more in-depth analysis considering different test cases will be shown at the conference.

I. INTRODUCTION

In recent years, significant attention has been given to extending Smoothed Particle Hydrodynamics (SPH) to model turbulent flows, as demonstrated by studies such as [1], [2], [3]. This advancement is critical for developing numerical methods suited for engineering applications but poses substantial theoretical and numerical challenges.

A promising approach to turbulence within the SPH framework builds on the principles of Large-Eddy Simulations (LES). As demonstrated in [4] and [5] LES relies on filtering the Navier–Stokes equations, a process closely aligned with the derivation of SPH's smoothed differential operators. In the framework of LES Dalrymple & Rogers [6] developed a Sub-Particle Scaling (SPS) scheme for modelling non-resolved eddies which was succesfully applied to wave breaking. More recently, analyses have been made on the discretization of wall boundary conditions in presence of turbulence [7], [8], but this remains a subject requiring further research, particularly at high Reynolds numbers such as river simulations [9]. Such cases require a very high resolution close to the boundary to correctly resolve the boundary layer which is difficult to introduce in SPH schemes.

In mesh-based method, the use of wall function is a standard approach to model the near-wall behaviour of turbulent flows, preventing the use of high resolution near boundaries but, to the best of the authors' knowledge, no investigation of wall functions have been made in the framework of SPH schemes. The present work has been developed using the open-source DualSPHysics code [10], which adopts the Sub-Particle Scale (SPS) formulation proposed by Dalrymple and Rogers [6] to model unresolved turbulence. Building on this foundation, we extend the mDBC approach of English et al. [11] for discretizing wall boundary conditions to compute wall shear stresses through a first-order consistent SPH spatial interpolation method, enhancing the accuracy of near-wall flow predictions. Furthermore, wall functions have been introduced within the SPH framework to efficiently model turbulent flow behavior near solid boundaries, significantly reducing the computational cost of resolving the boundary layer while maintaining reliable results.

II. NUMERICAL METHOD

This work has been developed in v5.4 of the open-source solver DualSPHysics [10] and uses the Laminar+SPS viscosity [6] option included in the solver with some modifications, as well as the mDBC boundary treatment [11] with the no-slip extensions and pressure formulation presented in [9]. When using the Laminar+SPS viscosity treatment the momentum equation takes the form

$$\frac{d\mathbf{u}_i}{dt} = -\sum_j m_j \left(\frac{P_i + P_j}{\rho_i \rho_j}\right) \nabla W_{ij} + \mathbf{g} \tag{1}$$

$$+ \sum_j m_j \left(\frac{4\nu \mathbf{r}_{ij} \cdot \nabla W_{ij}}{(\rho_i + \rho_j)(r_{ij}^2 + \eta^2)}\right) \mathbf{u}_{ij} + SPS \tag{2}$$

where $\mathbf{u}$ is particle viscosity, ρ density, m mass, and P pressure, Wij is the Wendland kernel for central particle i and neighbour particles j. $\mathbf{g}$ is the acceleration due to gravity, ν is the kinematic viscosity of the fluid and $\mathbf{r}_{ij} = \mathbf{r}_i - \mathbf{r}_j$ represents the difference between a quantity for a pair of particles, in this case position.

The sub-particle scale (SPS) viscosity takes the form

$$SPS = \sum_j m_j \left(\frac{\tau_i}{\rho_i^2} + \frac{\tau_j}{\rho_i^2} \right) \cdot \nabla W_{ij} \tag{3}$$

where τ is the SPS stress tensor defined as

$$\frac{\tau_i^{\alpha\beta}}{\rho_i^2} = \frac{1}{\rho_i} \left(2\nu_\tau \mathbf{S}_i^{\alpha\beta} - \frac{2\nu_\tau}{3} \mathbf{S}_i^{\gamma\gamma} \delta^{\alpha\beta} \right) + \frac{2}{3} \frac{C_I \Delta^2}{\rho_i} |S_i| \delta^{\alpha\beta} \tag{4}$$

where $C_I = 0.00066$, Δ is the initial particle spacing and $\mathbf{S}_i^{ab}$ is the rate of strain tensor defined as

$$\mathbf{S}^{\alpha\beta} = -\frac{1}{2} \left(\frac{\partial \mathbf{u}_\alpha}{\mathbf{x}_\beta} + \frac{\partial \mathbf{u}_\beta}{\mathbf{x}_\alpha} \right) \tag{5}$$

$|S_i| = (2\mathbf{S}_i^{\alpha\beta}\mathbf{S}_i^{\alpha\beta})^{1/2}$, and $\nu_{tau} = (C_s\Delta)^2|S_i|$ is the turbulent viscosity with Smagorinsky constant $C_s = 0.12$.

A. *Boundary particle Shear stresses*

In the current implementation of the Laminar+SPS model in DualSPHysics the equation for the SPS interaction, Equation 3, is only evaluated for fluid-fluid interaction. In the present approach this is extended to consider fluid-boundary interactions i.e. central fluid particle interacting with neighbouring boundary particles. In order to do this shear stress values are required for boundary particles. Shear stresses are found using a mDBC style approach with a Liu and Liu corrected kernel gradient summation to calculate the velocity gradient needed to build the rate of strain tensors. Corrected velocity sums and gradients are evaluated at ghost nodes g placed on the boundary surface (Figure 1) and summed over the surrounding fluid particles, the corrected velocity gradients are then found by solving the linear system, for the u velocity component for example,

$$\mathbf{A}_g \left[u_g; \partial u_g/\partial x; \partial u_g/\partial y; \partial u_g/\partial z \right] = \mathbf{b}_g \tag{6}$$

More details of the inversion process can be found in [9], [11]. When using a cartesian grid for boundary particles, this approaches results in columns of particles sharing the same ghost node, and therefore having the same values for velocity gradients and therefore shear stresses.

Once the velocity gradients have been evaluated they are used to create rate of strain tensors for the boundary particles following Equation 5. The stress tensor for the boundary particles is then found by

$$\frac{\tau_b^{\alpha\beta}}{\rho_b^2} = \frac{1}{\rho_b} \left(2(\nu + \nu_\tau) \mathbf{S}_b^{\alpha\beta} \right) \tag{7}$$

where the sum of the kinematic viscosity and turbulent viscosity is used as for wall functions. By considering only the components tangential to the wall the shear stresses at the wall can be found.

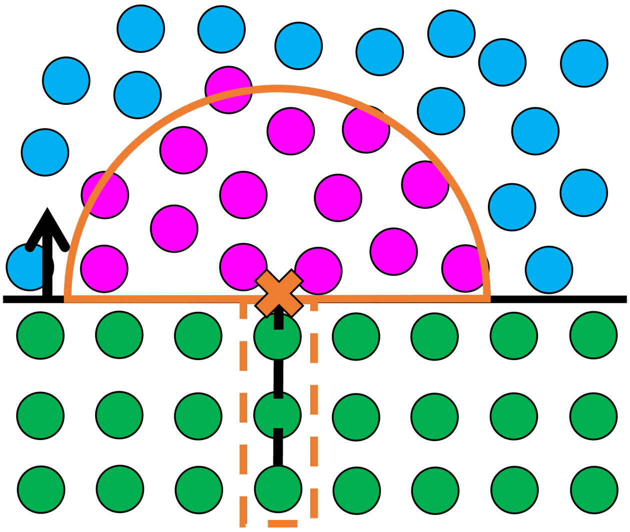

Fig. 1. mDBC corrected kernel and fluid particle neighbours for velocity gradient calculations.

B. *Wall Functions*

When modeling turbulent flows with grid based methods it is not possible to resolve the boundary layer fully, and the first grid cell may be located in the so called log-law region. When this happens the shear stresses are not accurately predicted resulting in poor results. To combat this wall functions are used to modify the viscosity for near boundary cells, altering the flow velocity and improving the approximation of the shear stresses. Following a standard wall function approach , in SPH the flow near a wall at a particle p can be characterised by two non-dimensional parameters, $Y^+ = y_p u_{tau}/\nu$ and $U^+ = u_p/u_{tau}$, where $u_\tau = \sqrt{\tau_w/\rho}$ is the so called friction velocity defined as the square root of the wall shear stress and y_p and u_p are the distance of the particle from a boundary and its stream wise velocity. These properties are related through the following relationship

$$U^+ = Y^+ \qquad Y^+ < 11.25 \tag{8}$$

$$U^+ = \frac{1}{\kappa} \log(EY^+) \qquad Y^+ > 11.25 \tag{9}$$

known as the viscous sub layer and log law regions respectively, where $\kappa = 0.4187$ is the von Karman constant and $E = 9.793$. An automatic wall function can be applied to satisfy these conditions by modifying the turbulent viscosity, $\nu_\tau = \nu_\tau + \nu_w$, of near wall particles using a wall viscosity given by

$$\nu_w = 0 \qquad Y^+ < 11.25 \tag{10}$$

$$\nu_w = \nu \left(\frac{Y^+ \kappa}{\log(EY^+)} - 1 \right) \qquad Y^+ > 11.25 \tag{11}$$

Finally, the value for of Y^+ for a particle can be found by iterating

$$\frac{u_p y_p \kappa}{\nu} - Y^+ \log(EY^+) = 0 \tag{12}$$

through, for example, a Newton-Raphson scheme using the definition of Y^+ and the value of τ_w found at the boundary particle as an initial guess.

In grid based methods wall functions are only applied to the first layer of nodes adjacent to a boundary as these are the only nodes to interact with the boundary nodes. Due to SPH using a smoothing kernel with support larger than one particle for its interactions, wall functions are here implemented for all fluid particles within a kernel radius of the boundary and therfore interact with boundary particles.

III. RESULTS

Presented here are the results for a turbulent channel flow simulated using SPH wall functions. The channel flow is the $Re = UL/\nu = 2857$ case defined in Lee and Moser [12]. The 3D channel is setup with periodic conditions in the x and y directions with half width $L = 1m$ in the z direction, maximum velocity $U = 1$ and viscosity $\nu = 0.00035$. In order to prevent period boundaries being too close and causing issues, but also in the interest of reducing run times the channel is made 5m in the x direction, and 0.4m in the y. The flow is initialised with the analytical solution for a laminar Poiseuille flow with the same flow properties and simulated for $100s$ to allow turbulence to develop. Two SPH particle sizes are simulated, $dx = 0.04m$ (10 particles wide channel) and $dx = 0.02m$ (20 particle wide channel) each with a smoothing length $h = 1.3dx$.

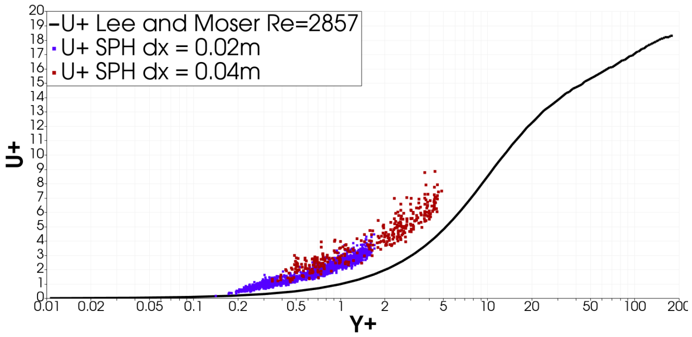

Fig. 2. Comparison of $Y+$ vs $U+$ results for two SPH resolutions (blue and red points) and numerical results of Lee and Moser [12] (black line)

Figure 2 shows the results for the two SPH resolutions with a selection of near boundary particle plotted with the coarser $dx = 0.04m$ in red and the finer $dx = 0.02m$ in blue at the final timestep. The SPH results are compared to the data of Lee and Moser [12] plotted in black. A the two SPH particle spacing result in different sized kernels the two sets of data spread over slightly different ranges of Y^+ values, but both staying in the viscous sub layer with $Y^+ < 11.25$. As SPH is a Lagrangian method the particles are able to move close to the boundary and therefore cover a larger than usual range of Y^+ values than seen grid based methods.

Overall, the SPH results follow the general shape and trend of the Lee and Moser results, however the is a discrepancy in the U^+ velocity values across the region. It appears the viscosity near the boundary is too low and therefore no enough resistance to flow is being applied. However, as a first attempt at incorporating wall functions in SPH the agreement in the shape of the profile is promising and warrants further investigation.

IV. CONCLUSIONS

In the present work the mDBC scheme for wall discretization has been extended for modelling turbulent flows near boundaries. A first order consistent formulation for computing wall shear stresses has been proposed, together with wall functions. Preliminary results demonstrate improved near-wall accuracy and reduced computational demands. This work highlights the potential of SPH for efficient and accurate simulations of turbulent flows. Further validation and analysis across different test cases will be essential to confirm the robustness and applicability of the approach in diverse engineering applications.

REFERENCES

[1] Violeau, D. and Issa, R., Numerical modelling of complex turbulent free-surface flows with the SPH method: an overview, International Journal for Numerical Methods in Fluids, 53, 2, pages 277-304, https://doi.org/10.1002/fld.1292

[2] J.J. Monaghan, A turbulence model for Smoothed Particle Hydrodynamics, European Journal of Mechanics - B/Fluids, 30, 4, 2011, 360-370, https://doi.org/10.1016/j.euromechflu.2011.04.002.

[3] Mayrhofer, A., Laurence, D., Rogers, B.D., and Violeau D., DNS and LES of 3-D wall-bounded turbulence using smoothed particle hydrodynamics, Comput. Fluids 115, 86–97 (2015).https://doi.org/10.1016/j.compfluid.2015.03.029

[4] A. Di Mascio, M. Antuono, A. Colagrossi, S. Marrone; Smoothed particle hydrodynamics method from a large eddy simulation perspective. Physics of Fluids 1 March 2017; 29 (3): 035102. https://doi.org/10.1063/1.4978274

[5] Antuono, M., Marrone, A., Di Mascio, A., Colagrossi A. Smoothed particle hydrodynamics method from a large eddy simulation perspective. Generalization to a quasi-Lagrangian model. Physics of Fluids 1 January 2021; 33 (1): 015102. https://doi.org/10.1063/5.0034568

[6] R.A. Dalrymple, B.D. Rogers, Numerical modeling of water waves with the SPH method, Coastal Engineering, Volume 53, Issues 2–3, 2006, Pages 141-147, ISSN 0378-3839, doi:10.1016/j.coastaleng.2005.10.004.

[7] Laha, S., Fourtakas, G., Das, P.K. et al. Towards the estimation of wall shear stress in smoothed particle hydrodynamics. Comp. Part. Mech. (2024). https://doi.org/10.1007/s40571-024-00879-9

[8] Wang, F., Wu, D., Hu, X. SPH formulations for RANS modeling. Proceedings of 18th SPHERIC workshop (2024), pages 207 - 2014, ISBN 978-3-00-079208-3.

[9] English, Aaron and Vacondio, Renato and Dazzi, Susanna and Domínguez, Jose M., River Flows Past Bridges: A Comparison of 3d Smoothed Particle Hydrodynamics and 2d Shallow Water Equations Models. Available at SSRN: https://ssrn.com/abstract=5063848 or http://dx.doi.org/10.2139/ssrn.5063848

[10] Domínguez, J.M., Fourtakas, G., Altomare, C. et al. DualSPHysics: from fluid dynamics to multiphysics problems. Comp. Part. Mech. 9, 867–895 (2022). https://doi.org/10.1007/s40571-021-00404-

[11] English, A., Domínguez, J.M., Vacondio, R. et al. Modified dynamic boundary conditions (mDBC) for general-purpose smoothed particle hydrodynamics (SPH): application to tank sloshing, dam break and fish pass problems. Comp. Part. Mech. 9, 1–15 (2022). https://doi.org/10.1007/s40571-021-00403-3

[12] Lee M, Moser RD. Direct numerical simulation of turbulent channel flow up to $Re_\tau \approx 5200$. Journal of Fluid Mechanics. 2015;774:395-415. doi:10.1017/jfm.2015.268