Smoothed Particle Hydrodynamics Simulation of 3D Open Channel Flow Over Vegetated Bed

Efstathios Chatzoglou
Department of Civil Engineering, Hydromechanics
and Environmental Engineering Laboratory,
University of Thessaly
Volos, Greece
efchatzoglou@uth.gr

Antonios Liakopoulos
Department of Civil Engineering, Hydromechanics
and Environmental Engineering Laboratory,
University of Thessaly
Volos, Greece
aliakop@uth.gr

I. INTRODUCTION

Vegetation significantly influences turbulent flow in natural open channels. It also impacts sediment transport, which affects the functioning of wetlands and floodplains [1]. The presence of aquatic plants increases flow resistance and reduces the average velocity compared to areas without vegetation. Understanding the relationship between vegetation type and flow characteristics is crucial for the planning and design of flood control systems or eco-hydraulic research [2,3]. In a similar study, Kasiteropoulou et al. [4] employed ANSYS-CFX for the simulation of the same problem. Their results were compared to experimental studies and prior simulations from the literature.

This study examines turbulent flow in an open channel with vegetated bed. Vegetation is modelled as a series of small-diameter rigid cylinders placed vertical to the channel bed. We use the Smoothed Particle Hydrodynamics (SPH) method to simulate the complex fluid-vegetation interaction. Unlike traditional computational methods, such as the finite volume approach, the SPH method eliminates the need for meshing, handling fluid dynamics using computational nodes (particles). While traditional methods have demonstrated accuracy, SPH offers superior handling of irregular geometries and complex open channel flows [5]. SPH provides better treatment of irregular bed topography and vegetation structures. Recent research has shown the applicability of SPH to similar problems with promising results [6-8].

This study evaluates the performance of the SPH method by comparing it with traditional 3-D computations based on finite volume methods and experimental studies conducted by other researchers. For the simulations we use the DualSPHysics code [9]. All simulations run on an NVIDIA RTX 3060 GPU card. Results demonstrate that the SPH method provides accurate results, capturing detailed flow patterns and interactions with the idealized vegetation.

II. COMPUTATIONAL MODEL AND SOLUTION METHODOLOGY

Developing a vegetation model requires selecting several parameters, including the shape, height, vegetation density, and flexibility (or rigidity) of the plants. Dunn et al. [10] conducted a series of laboratory experiments to study channel flow with a free surface and calculate the resistance caused by bed vegetation. These experiments were performed in a flume with a slope ranging from 0.0036 to 0.0161. The flume was 19.5 m in length, 0.91 m in width, and 0.61 m in depth. Flexible and rigid cylinders were used to simulate vegetation.

To compare our SPH results with the experimental work of Dunn et al. [10] and the numerical simulations of Kasiteropoulou et al. [4], we kept the same geometry and modelled vegetation using rigid cylinders. While this neglects plant flexibility, it establishes a baseline for understanding flow resistance caused by vegetation. Unlike the experiments, we used periodic boundary conditions in the streamwise direction. Thus, a shorter computational domain length was used in this study, compared to the 19.5 m channel in the experiments, to reduce computational cost. Details of the model's geometry and parameters are summarized in Table 1. Fig. 1 illustrates a top view of the computational domain. Notice the staggered arrangement of the cylinders, which mimics natural vegetation and introduces flow heterogeneity and turbulence typically observed in vegetated channels. For the simulation of the solid boundary, we applied the modified dynamic boundary conditions (mDBCs) proposed by English et al. [11] to ensure as accurate as possible fluid/solid particle interactions.

TABLE 1. SIMULATION PARAMETERS. L IS THE CHANNEL'S LENGTH AND B IS THE CHANNEL'S WIDTH.

Geometry	**L (m)**	**B (m)**	**Flow depth (m)**	**cylinder radius (m)**	**cylinder height (m)**
	0.25	0.91	0.233	0.00175	0.1175
Parameters	**Streamwise dir. BCs**	**Liquid/Solid BCs**	**Turbulence model**	**Viscosity (m^2/s)**	**Density (Kg/m^3)**
	Periodic	mDBCs	Laminar+SPS	0.000001	1000

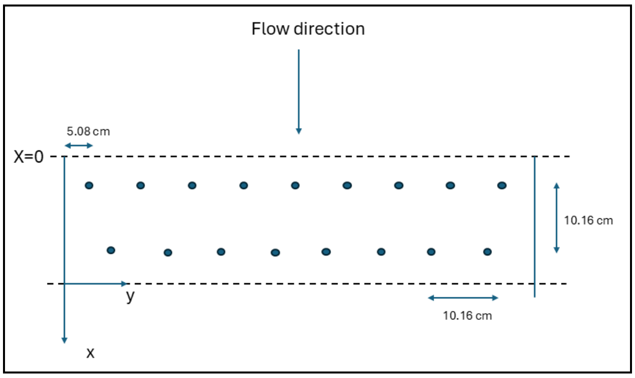

Figure 1. Top view of the computational domain showing the staggered cylinder arrangement.

III. RESULTS AND DISCUSSION

A contour plot of the X-component of velocity at the vertical plane X = 0.125 m is shown in Fig. 2. Near the solid walls, the velocity profile undergoes significant changes, with sharply reduced velocities observed close to the walls and in the locations where the cylinders are placed. The rigid cylinders significantly affect the velocity pattern.

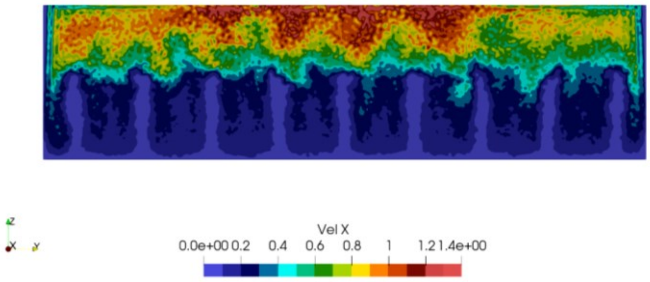

Figure 2. Side view. X-component of velocity on channel's plane X=0.125 m, T= 200 sec.

Mass conservation law is satisfied, meaning that no particles escape through the solid boundaries. The volume flow rate is calculated via the "FlowTool" postprocessing tool of the DualSPHysics code (Q = 0.181 m^3/s). The mean velocity is V=0.85 m/s. These parameter values are in accordance with the experimental work of Dunn et al. [10] and the simulation results of Kasiteropoulou et al. [4].

The SPH space averaged velocity profile from various spanwise locations (Y = 0.01–0.9 m) is depicted in Fig. 3. The comparison shows good agreement between the SPH results and the CFX results from Kasiteropoulou et al. [4]. Moreover, the SPH model demonstrates even better agreement with the measurements of Dunn et al. [10], indicating that SPH is a promising tool for simulating such flows. This accuracy can be attributed to SPH's ability to model free-surface dynamics and small-scale turbulence effectively.

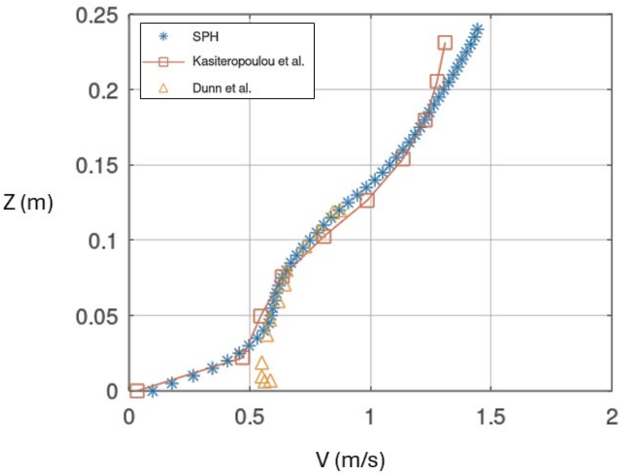

Figure 3. Comparison of velocity profiles obtained from the SPH model with numerical results by Kasiteropoulou et al. [4] and experimental data by Dunn et al. [10].

IV. CONCLUSION

This work demonstrates the ability of the SPH method to simulate real-life engineering problems and produce reliable results, even outperforming conventional computational fluid dynamics (CFD) methods. The authors' future goals include using the project CHRONO [9] to incorporate flexibility into the cylinders, thereby making the problem more representative of real-world conditions. In addition, an in-house implementation of the Dynamic Smagorinsky turbulence model in the DualSPHysics source code will be tested on the same problem.

ACKNOWLEDGMENT

This research project was supported by the Hellenic Foundation for Research and Innovation (H.F.R.I.) under the "2nd Call for H.F.R.I. Research Projects to support Faculty Members & Researchers" (Project Number: 4584). E. Chatzoglou was supported by a doctoral candidate scholarship of the Center of Research Innovation and Excellence of the University of Thessaly, funded by the Special Account for Research Grants of the University of Thessaly.

References

[1] Curran, J. C., & Hession, W. C. "Vegetative impacts on hydraulics and sediment processes across the fluvial system", Journal of Hydrology, 505, 364-376, 2013.

[2] Nepf, H., & Ghisalberti, M. "Flow and transport in channels with submerged vegetation", Acta Geophysica, 56, 753-777, 2008.

[3] Zhou, J., Fu, B., Gao, G., Lü, Y., Liu, Y., Lü, N., & Wang, S. "Effects of precipitation and restoration vegetation on soil erosion in a semi-arid environment in the Loess Plateau, China", Catena, 137, 1-11, 2016.

[4] Kasiteropoulou, D., Liakopoulos, A., Michalolias, N., & Keramaris, E. "Numerical modelling and analysis of turbulent flow in an open channel with submerged vegetation", Environmental Processes, 4, 47-61, 2017.

[5] Violeau, D. "Fluid mechanics and the SPH method: theory and applications", Oxford University Press, 2012.

[6] Martínez-Estévez, I., Tagliafierro, B., El Rahi, J., Domínguez, J. M., Crespo, A. J., Troch, P., & Gómez-Gesteira, M. "Coupling an SPH-based solver with an FEA structural solver to simulate free surface flows interacting with flexible structures", Computer Methods in Applied Mechanics and Engineering, 410, 115989, 2023.

[7] O'Connor, J., & Rogers, B. D. "A fluid–structure interaction model for free-surface flows and flexible structures using smoothed particle hydrodynamics on a GPU", Journal of Fluids and Structures, 104, 103312, 2021.

[8] Wang, L., et al. "Numerical investigation on solitary waves traveling over rigid vegetation by a 3D-MPS method", Applied Ocean Research, 132, 103476, 2023.

[9] J.M Domínguez, G. Fourtakas, Altomare C., et al. "DualSPHysics: from fluid dynamics to multiphysics problems", Comp. Part. Mech., vol. 9, pp. 867–885, 2022.

[10] Dunn, C., Lopez, F., & Garcia, M. "Mean flow and turbulence in a Laboratory Channel with simulated vegetation", Hydrosystems Laboratory, Department of Civil Engineering, University of Illinois, Urbana-Champaign, 1996.

[11] English, A., Domínguez, J. M., Vacondio, R., Crespo, A. J. C., Stansby, P. K., Lind, S. J., & Gómez-Gesteira, M. "Modified dynamic boundary conditions (mDBC) for general-purpose smoothed particle hydrodynamics (SPH): application to tank sloshing, dam break and fish pass problems", Computational Particle Mechanics, 9(5), 1-15, 2022.