A NESTING STRATEGY TO MODEL A PROPELLER JET WAKE ON A SCOURED BATHYMETRY USING SPH

D. Ferraro, R. Gaudio, F. Aristodemo
Department of Civil Engineering
University of Calabria
Rende, Italy
domenico.ferraro@unical.it

J. M. Domínguez
Environmental Physics Laboratory CIM-UVIGO
University of Vigo
Vigo, Spain
e-mail address if desired

A. Lauria
Department of Engineering for Innovation
University of Salento
Lecce, Italy
e-mail address if desired

C. Altomare
Maritime Engineering Laboratory LIM/UPC
Polytechnic University of Catalonia
Barcelona, Spain
e-mail address if desired

This study investigates the 3D hydrodynamics of a marine propeller jet (PJ) operating over a Scoured Bathymetry (SB). The challenge lies in simulating both the small-scale, high-resolution dynamics of the propeller jet and the larger-scale experimental domain. To achieve this, the authors developed a nesting strategy within the Smoothed Particle Hydrodynamics (SPH) framework. This method allows for coupling high-resolution simulations of propeller hydrodynamics within a localized area to a larger, lower-resolution domain, representative of a laboratory flume setup. The results were validated through physical experiments and emphasize that SPH is able to simulate complex 3D hydrodynamics. The nesting methodology is highlighted as a valuable tool for future studies on fluid-structure interactions.

I. Introduction

The role of understanding maritime propeller hydrodynamics is crucial for optimizing marine vessel performance, fuel efficiency, and environmental impact. Over recent decades, advances in experimental techniques and Computational Fluid Dynamics (CFD) have provided valuable insights into the flow behaviour of maritime propellers, which are often used instead of more time-consuming experimental investigation and costly measurement facility. In the meanwhile, CFD techniques have evolved, offering detailed analyses of flow patterns and vortex structures, that are hard to capture in experiments. Particular attention was paid to the propeller hydrodynamics in the near region (e.g. [1] among others). Several studies related to the near field of the PJ wake were conducted with Large Eddy Simulation (LES) [2] or Detached Eddy Simulations (DES) [3], whereas more recent research, like that by [4], explored the hydrodynamics of the PJ by using a Reynolds-Averaged Navier-Stokes (RANS) approach. Nevertheless, numerical simulations, such as the RANS method, have become widely used, but are still limited by dissipation effects, when predicting global variables. While Finite Volume Methods (FVMs) dominate current numerical simulation approaches, Lagrangian methods, such as Smoothed Particle Hydrodynamics (SPH), have received less attention. Only a few studies, like [5], have attempted to model propeller wakes using SPH. However, these analyses were focused on related phenomena like bed erosion rather than the hydrodynamics of the propeller itself. As a result, the fluid dynamics around ship propellers remains challenging to be simulate, particularly using Lagrangian methods, which demand significant computing resources.

II. Laboratory tests

The experiments took place at the *Laboratorio "Grandi Modelli Idraulici"* at the University of Calabria (Italy) in a tilting flume with a rectangular cross-section measuring 0.985 m in width, 0.8 m in depth, and 16 m in length. A 2.5m-long and 0.30m-deep recess box was positioned 10 m from the flume inlet. The recess box was filled with a uniform fine sand having median diameter $d_{50} = 0.69$ mm and specific sediment weight γ_s=2660 kg/m^3. A four-blade propeller, depicted in Fig. 1(a), operates at a rotational speed of $n = 700$ rpm and has a diameter $D_P = 6$ cm with a thrust coefficient $C_t = 0.5$. Additional details about the propeller system can be found in the work of [6]. The efflux velocity U_0 was computed using the formula $U_0 = 1.22 n^{1.01} D_P^{0.84} C_t^{0.62} = 0.89$ m/s [7].

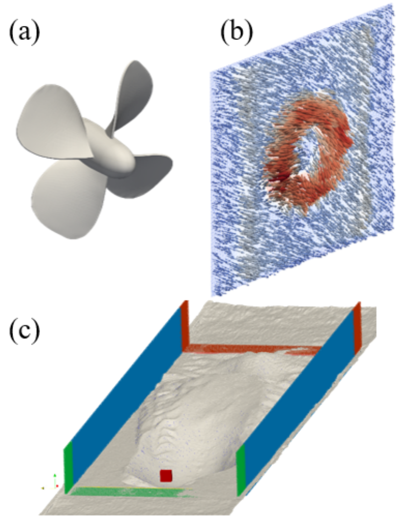

Fig. 1. (a) Stereolithography (STL) model of the propeller, (b) snapshot of the gauge mesh containing 3D velocity vectors, (c) STL model of the SB with boundary conditions positions.

The test conditions are summarized as follows. The flow discharge was $Q = 23.2$ l/s, the water depth over the initial bed $h_w = 16.0$ cm, the bulk Reynolds number $Re = 4RU/\nu = 5.43 \cdot 10^4$, $R = 0.121$ being the hydraulic radius and $\nu = 1.31 \cdot 10^{-6}\mathrm{m}^2\mathrm{s}^{-1}$ the kinematic viscosity of water at 10°C, the longitudinal bottom slope $S = 0.3\%$, the Froude number $Fr = U/\sqrt{gh_w} = 0.135$ and the gravity acceleration $g = 9.806$ m/s^2. For further details, see [8].

The laboratory test was initiated once the channel flow rate was achieved, and the propeller was switched on. The test was stopped when the scour produced by the PJ no longer evolved significantly, indicating that the Quasi-Equilibrium Scour (QES) condition had been reached [6]. Upon reaching the QES, measurements of the 3D instantaneous velocity components (streamwise, u, spanwise, v, and vertical, w) were taken using a four-beam down-looking Acoustic Doppler Velocimeter (ADV) probe with a sampling rate of 100 Hz and a sampling duration of 600 seconds. These measurements were recorded at equispaced points along the PJ wake (horizontal coordinate of the propeller axis). Finally, the QES bathymetry was acquired using photogrammetry, as detailed in [6]. The resulting digital terrain model was further processed to obtain a STL model for use in numerical simulations.

III. DUALSPHYSICS MODEL

DualSPHysics solver is an implementation of the SPH method optimised for execution on parallel hardware such as multicore central processing units (CPUs) and NVIDIA graphics processing units (GPUs). Its high parallelisation makes it possible to tackle large simulations in reasonable times. In addition, DualSPHysics includes coupling with other models such as DEM, Project Chrono and MoorDynPlus, that enable multiphysics simulations. More details of the SPH formulation included in DualSPHysics can be found in [9]. Finally, the LES Sub-Particle Scale (SPS) model [10] was used and the dissipation term was mirrored by the water laminar viscosity with SPS.

A. Boundary conditions

This work employs modified Dynamic Boundary Conditions (mDBC) [11]. They were applied to the QES bathymetry. This process results in smoother, more realistic pressure fields by treating boundary density as part of a fluid continuum, eliminating the non-physical gaps present with standard DBC. The ratio between two viscosity parameters was taken into account to simulate the interaction among fluid particles and boundary particles by ”ViscoBoundFactor” (VBF) in DualSPHysics code. This factor, applied to the QES bathymetry, mimics the roughness effect of the sand bed. Since it is a multiplier of the fluid viscosity, which is the real viscosity value in the Laminar+SPS model, the value of the VBF was found by a physically based procedure. Hence, considering the Blasius friction factor formula, $f_B = 0.316 * Re^{-0.25}$, which depends on the fluid viscous property, and the Manning formula, $f_M = k^{1/3}/R^{1/3}$, depending on the roughness height ($k_s \approx d_50$), one can define a $VBF = f_M/f_B \approx 1100$.

Finally, to simulate a steady flow, boundary conditions were applied at both the inlet and outlet in the current model, as bounded by the green and red wide lines in Fig. 1(c). The algorithm uses buffer layers to discretize the open boundaries, enabling the assignment of flow variables [12]. In particular, at the inlet the bulk velocity and refilling particles below the assigned water depth elevation were imposed, whereas extrapolating values from within the domain were setup for the outlet.

B. Nesting strategy

Integrating the results of a high-resolution numerical simulation of PJ dynamics into a larger-scale domain is a challenging issue. Propeller geometries have sub-millimeter dimensions, which play a critical role in determining the appropriate resolution for the numerical simulation which becomes prohibitively expensive and time-consuming. For instance, capturing the minute details of the propeller blade, which has a thickness of about 1 mm, requires at least a particle dimension $d_p = 0.3$ mm and a small smoothing length $h = c_h\sqrt{3d_p^2}$, where $c_h = 1.0$, to avoid interaction between upstream and downstream particle interspersed by the blade. Thus, such simulations demand fine meshing and significant processing power, which limit the applicability of these models to real, or even, laboratory scales. While high-resolution simulations are crucial for understanding localized flow dynamics around the propeller, applying such detail across a multi-meter SB domain would result in computational overload and impractical runtimes. To overcome this limitation, we develop a nesting strategy within the SPH framework, which enables to capture and store the 3D velocity field in a target region of the high-resolution simulations in confined, localized domains (see Fig. 1,b). Then, the sampled 3D velocity field was nested in a larger, lower-resolution domains that represent the overall experimental setup to simulate hydrodynamic features far from the propeller. The method used to transfer the fluid velocity field from one simulation to another can be understood by referring to the MESH-IN technique [13]. In contrast, in the present study, the pre-recorded velocity field is applied directly to a fluid region rather than to the simulation inlet boundaries. This approach specifically targets the area most influenced by the propeller. The velocity field is introduced through a relaxation zone technique, where the velocity imposition varies according to the distance from the propeller, following a hyperbolic function, as detailed in [14].

IV. Results and Conclusions

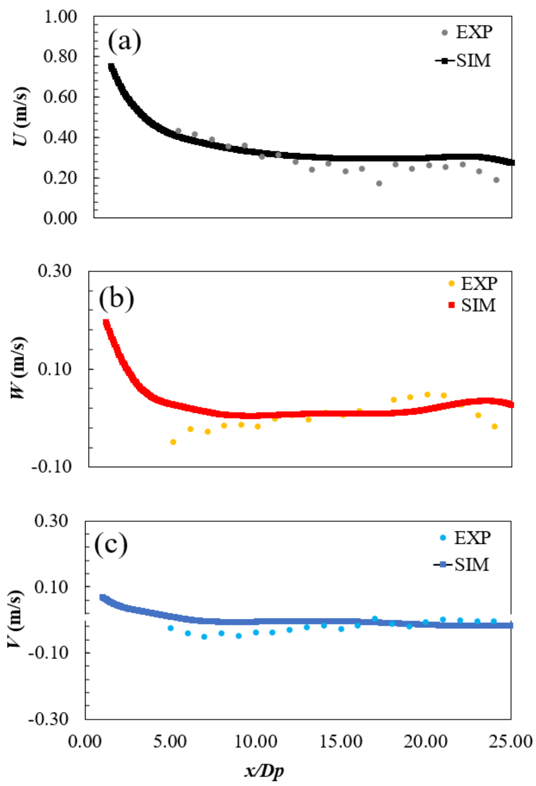

Fig. 2. Panels (a), (b), and (c) contain the comparison between laboratory data and simulation results of time-averaged streamwise, spanwise, and vertical velocity, respectively.

To validate the SPH results, time-averaged velocity profiles were compared with those measured using the ADV, demonstrating satisfactory agreement. In Fig. 2, the key flow characteristics in the measured sections are clearly visible. Specifically, Figs. 2(a), (b), and (c) present the spatial variation of streamwise, spanwise, and vertical time-averaged velocities (U, V, and W) downstream of the propeller, comparing experimental data and SPH results. In the region near the propeller, the numerical model tends to overpredict U, especially where the flow exhibits high gradients. This overprediction was also noted in [8], where a RANS model was applied. However, there is strong agreement between experiments and simulation for $5.0 \leq x/D_p \leq 20$. Discrepancies are also observed in the V and W components for $x/D_p \leq 5.0$, likely due to the challenges posed by the high-velocity gradients in this region. Nonetheless, the SPH model accurately represents V and W for $x/D_p \geq 5.0$.

In conclusion, this work presents a novel application of SPH to model PJ dynamics over SB, addressing complexities previously underexplored. The integration of high-resolution simulations in a larger-scale domain enhances understanding of fluid-structure interactions in marine environments. While SPH can be more intuitive for modeling complex fluid dynamics, it may require expertise and longer setup times for non-specialist engineers. Compared to traditional methods, SPH excels in simulating free-surface flows and interactions, providing insights that conventional techniques struggle to capture, although it may demand significant computational resources. Overall, the simulated velocity profiles exhibit satisfactory agreement with the experimental data, although some discrepancies remain at specific locations, particularly in the near-propeller flow region, which is crucial for understanding the flow field evolution of a PJ wake.

References

[1] Albertson, M. L., Dai, Y., Jensen, R. A., Rouse, H. "Diffusion of submerged jets." Transactions of the American Society of Civil Engineers 115.1, 639-664, 1950.
[2] Kumar, P., and Krishnan M. "Large eddy simulation of propeller wake instabilities." Journal of Fluid Mechanics 814, 361-396, 2017.
[3] Qin, D., Huang, Q., Pan, G., Han, P., Luo, Y., Dong, X. "Numerical simulation of vortex instabilities in the wake of a preswirl pumpjet propulsor." Physics of Fluids 33.5, 2021.
[4] Wei, M., Yee-Meng C., and Nian-Sheng C. "Recent advances in understanding propeller jet flow and its impact on scour." Physics of Fluids 32.10, 2020.
[5] Ulrich, C., Leonardi, M., Rung, T. "Multi-physics SPH simulation of complex marine-engineering hydrodynamic problems." Ocean Engineering 64, 109-121, 2013.
[6] Ferraro, D., Lauria, A., Penna, N., Gaudio, R. "Temporal development of unconfined propeller scour in waterways." Physics of Fluids 33.9, 2021.
[7] Hamill, G., Kee, C., Ryan, D. "Three-dimension efflux velocity characteristics of marine propeller jets." Proceedings of the Institution of Civil Engineers-Maritime Engineering. Vol. 168. No. 2. Thomas Telford Ltd, 2015.
[8] Ferraro, D., Lauria, A., Penna, N., Gaudio, R. "Unconfined propeller scour in waterways: The role of flow intensity." Physics of Fluids 34.9, 2022.
[9] Domínguez, J.M., Fourtakas, G., Altomare, C. et al. DualSPHysics: from fluid dynamics to multiphysics problems. Comp. Part. Mech. 9, 867–895 (2022).
[10] Dalrymple, R. A., Rogers, B. D. "Numerical modeling of water waves with the SPH method." Coastal engineering 53.2-3, 141-147, 2006.
[11] English, A., et al. "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.
[12] Tafuni,A., Domínguez, J.M., Vacondio, R., Crespo, A.J.C. "A versatile algorithm for the treatment of open boundary conditions in Smoothed particle hydrodynamics GPU models." Computer Methods in Applied Mechanics and Engineering 342, 2018.
[13] Ruffini, G., Domínguez, J.M. et al. "MESH-IN: A MESHed INlet offline coupling method for 3-D extreme hydrodynamic events in DualSPHysics." Ocean Engineering 268, 2023.
[14] Altomare, C., Tagliafierro, B., Dominguez, J.M., Suzuki, T., Viccione, G. "Improved relaxation zone method in SPH-based model for coastal engineering applications." Applied Ocean Research 81, 2018.