water

MDPI

Editorial

Computational Fluid Mechanics and Hydraulics

Ahmad Shakibaeinia [1,*] and Amir Reza Zarrati [2]

[1] Department of Civil, Geological and Mining Engineering, Polytechnique Montreal, Montreal, QC H3T 1J4, Canada
[2] Department of Civil and Environmental Engineering, Amirkabir University of Technology, Tehran 1591634311, Iran; zarrati@aut.ac.ir
* Correspondence: ahmad.shakibaeinia@polymtl.ca

1. Introduction

Rapid advances in computational power and numerical techniques in recent years have provided us with the opportunity to solve challenging problems in many science and engineering fields. Fluid mechanics and hydraulics are no exception. The majority of computational approaches in fluid mechanics and hydraulics are physics-based, using numerical techniques to solve the governing equation of the physical phenomena, often in the form of partial differential equations (PDEs), such as Navier–Stokes, turbulnce models, and the advection–diffusion equation. Traditional numerical techniques such as the finite difference method (FDM), finite volume method (FVM), and finite element method (FEM) are based on the Eulerian formulation and rely on a mesh system over which the governing equations are discretized and solved. A newer generation of numerical techniques removes the mesh dependency and often applies a Lagrangian formulation. These so-called mesh-free Lagrangian (particle) methods are based on the motion of free-to-move particles, which makes them powerful and natural in the simulation of extreme deformations and fragmentations in fluid boundaries and interfaces. Smoothed particle hydrodynamics (SPH), moving-particle semi-implicit (MPS), and Lattice Boltzmann (LBM) methods are some of the widely used particle methods in continuum mechanics. Lagrangian methods for discrete simulations, such as the discrete element method (DEM), can also be categorized as particle methods. The simulation of modern fluid mechanics and hydraulics problems with the desired accuracy, especially those involving phenomena at multiple length and time scales, requires multi-million computational elements, resulting in a prohibitive simulation time. The growing high-performance computing (HPC) infrastructure, such as CPUs and GPUs, and massively parallel algorithms, has enabled handling such computational loads.

Citation: Shakibaeinia, A.; Zarrati, A.R. Computational Fluid Mechanics and Hydraulics. *Water* **2022**, *14*, 3985. https://doi.org/10.3390/w14243985

Received: 15 November 2022
Accepted: 1 December 2022
Published: 7 December 2022

In addition to the physics-based computational methods, the emerging data-driven approaches powered by modern machine learning (ML) techniques are increasingly finding their way into fluid simulations, as replacements or enhancements. A drawback of the data-driven approaches is that they are unaware of physics, leading to challenges with physical consistency and interpretability. To deal with this issue, the the recent research focuses on physics-informed ML techniques.

This Special Issue of *Water* focuses on computational aspects of hydraulics and fluid mechanics research. It aims to present and discuss the latest advancements in the numerical techniques (physics-based and data-driven) and their application for the simulation of environmental fluid mechanics and hydraulics problems, especially in natural or human-made hydro-systems (e.g., rivers and hydraulic structures).

2. Overview of This Special Issue

This Special Issue includes eleven original contributions, including nine research articles [1–9] and three review papers [10,11]. In terms of the modeling techniques, these papers develop and apply a variety of methods such as mesh-based Eulerian [1–3], mesh-free Lagrangian [4,5,7,8], and data-driven [6] techniques. These models are applied to study different hydraulics problems, including curved channels [1,2,11], channel confluences [10], air-core vortices [5], and submerged granular slides [3].

The article by Wang et al. [1] studies the turbulent flow pattern at a curved channel (with a 135-degree bend), under the influence of negatively buoyant jets, using a three-dimensional (3D) FVM. Numerical simulation results using three turbulence models (the standard and non-linear $k-\epsilon$ and the $k-\omega$ SST) are evaluated in comparison with results from the experiments conducted by the authors. In this study, the $k-\omega$ SST model shows the best performance. Strong secondary current cells are observed at the bend, but they are disturbed by the presence of the negatively buoyant jets.

The focuses of the article by Zhang et al. [2] are on the simulation of laminar flow in curved channels using a 3D FVM with a non-staggered triangular grid. It proposes and evaluates a filtering technique, which is proven to be effective in improving numerical accuracy and convergence. The model is applied to a sharply curved channel with a 180-degree bend, Reynolds (Re) number of 125, and different Froude (Fr) numbers. The Froude number is found to impact the pattern and number of secondary flow cells in the bend.

The article of Shademani et al. [3] couples a two-phase (gas-liquid) FVM with a discrete-element method (DEM) for 3D unresolved simulation of dense granular collapses in dry and submerged conditions. It also performs complementary experiments to provide data for validation and analysis. The comparison of numerical and experimental results demonstrates the accuracy of the developed numerical model. The initial granular column aspect ratio is found to have a significant impact on the collapse mechanism and the morphological evolution of granular material. The spatio-temporal variation of the volume fraction was also quantified.

The article by Feng et al. [4] improves the stability of the MPS particle method using a new anti-clustering technique. Evaluation and validation for different 2D and 3D benchmark cases show the effectiveness of the developed technique. The 3D MPS method is also successfully coupled with FVM for modeling of two-phase (liquid–gas) flows.

The study by Azarpira et al. [5] compares Eulerian and Lagrangian numerical approaches (based on the FVM and SPH methods, respectively) for the simulation of air-core vortices. Experiments are also performed to provide the validation data. FVM and SPH show comparable results, with SPH having a higher computational cost.

The paper by Cheng et al. [6] is based on a data-driven and machine-learning approach. It proposes a Physics-Informed Neural Network (PINN) combined with Residual Network (Resnet) blocks for the prediction of fluid flow fields. To include the physics, the neural network is constrained by the governing partial differential equation, embedded in the loss function. The developed PINN–Resnet model, evaluated for different benchmark cases, shows a better performance than the traditional deep learning techniques. PINN-Resnet also shows better accuracy than PINN for Navier–Stokes predictions.

The aim of the article by Eiris et al. [7] is to extend the prior work of its authors in introducing an SPH stabilization technique for ideal gasses to the case of incompressible viscous flows. It is based on SPH and an Arbitrary Lagrangian–Eulerian (ALE) framework. The performance of the proposed model is evaluated for several benchmark cases, such as Taylor–Green, Poiseuille flow, and lid-driven cavity. The model results show higher accuracy and smaller pressure oscillations compared to the other SPH methods from the literature.

The article by Krimi et al. [8] proposes an improved δ-SPH particle method with an automatic adaptive numerical dissipation. The developed approach is evaluated for different benchmark cases, including dam-break, Taylor–Green, and lid-driven cavity flows. The results demonstrate the accuracy of the proposed model without challenges associated with parameter dependency of the original δ-SPH method.

The article by Zhang et al. [9] studies near-bed turbulent flow structure by modeling open channel flow with square bars placed at the channel bed representing roughness elements. It uses FVM with large eddy simulation (LES) turbulence closure. The model is validated in comparison with experimental data. The study shows the important impact of spacing between bars on turbulence characteristics, such as turbulence intensity and shape and size of eddies (in the cavity region).

Two articles by Shaheed et al. [10,11] review recent progress on numerical simulations of secondary flows in two common morphological features of river channels, i.e., confluences and bends. The structure of the main flow features identified by the past literature, particularly the secondary flows, is discussed. The advantages and disadvantages of different numerical modeling techniques and turbulence models are also summarized.

Funding: This research received no external funding.

Data Availability Statement: Not applicable.

Acknowledgments: We would like to thank all the authors who contributed to this Special Issue. We would like to also thank all the reviewers for providing their valuable comments, which greatly improved the quality of the published papers.

Conflicts of Interest: The authors declare no conflict of interest.

References

1. Wang, X.; Mohammadian, A.; Rennie, C.D. Influence of Negatively Buoyant Jets on a Strongly Curved Open-Channel Flow Using RANS Models with Experimental Data. *Water* **2022**, *14*, 347. [CrossRef]
2. Zhang, W.; Uh Zapata, M.; Pham Van Bang, D.; Nguyen, K.D. Three-Dimensional Hydrostatic Curved Channel Flow Simulations Using Non-Staggered Triangular Grids. *Water* **2022**, *14*, 174. [CrossRef]
3. Shademani, M.; Blais, B.; Shakibaeinia, A. CFD-DEM Modeling of Dense Sub-Aerial and Submerged Granular Collapses. *Water* **2021**, *13*, 2969. [CrossRef]
4. Feng, M.; Huang, S.; Lian, G. An Anti-Clustering Model for Stability Enhancement of a 3D Moving Particle Semi-Implicit Method and Two-Phase Coupling between MPS and Euler Grids. *Water* **2021**, *13*, 887. [CrossRef]
5. Azarpira, M.; Zarrati, A.R.; Farrokhzad, P. Comparison between the Lagrangian and Eulerian Approach in Simulation of Free Surface Air-Core Vortices. *Water* **2021**, *13*, 726. [CrossRef]
6. Cheng, C.; Zhang, G.-T. Deep Learning Method Based on Physics Informed Neural Network with Resnet Block for Solving Fluid Flow Problems. *Water* **2021**, *13*, 423. [CrossRef]
7. Eirís, A.; Ramírez, L.; Fernández-Fidalgo, J.; Couceiro, I.; Nogueira, X. SPH-ALE Scheme for Weakly Compressible Viscous Flow with a Posteriori Stabilization. *Water* **2021**, *13*, 245. [CrossRef]
8. Krimi, A.; Ramírez, L.; Khelladi, S.; Navarrina, F.; Deligant, M.; Nogueira, X. Improved δ-SPH Scheme with Automatic and Adaptive Numerical Dissipation. *Water* **2020**, *12*, 2858. [CrossRef]
9. Zhang, Z.; Li, S.S. Large Eddy Simulation of Near-Bed Flow and Turbulence over Roughness Elements in the Shallow Open-Channel. *Water* **2020**, *12*, 2701. [CrossRef]
10. Shaheed, R.; Yan, X.; Mohammadian, A. Review and Comparison of Numerical Simulations of Secondary Flow in River Confluences. *Water* **2021**, *13*, 1917. [CrossRef]
11. Shaheed, R.; Mohammadian, A.; Yan, X. A Review of Numerical Simulations of Secondary Flows in River Bends. *Water* **2021**, *13*, 884. [CrossRef]