

TWO-PHASE EULER MODELLING OF HYDRAULIC STRUCTURES

LOURENÇO SASSETTI MENDES¹, ANTÓNIO MURALHA²

¹*Laboratório Nacional de Engenharia Civil, Lisbon, Portugal, lmendes@lnec.pt*

²*Laboratório Nacional de Engenharia Civil, Lisbon, Portugal, amuralha@lnec.pt*

Keywords: *complete Euler-Euler, hydraulic structures, energy dissipation, aeration*

Resume

The Two-Phase Euler model, implemented in the twoPhaseEulerFoam solver of OpenFOAM v2412, is applied to flows within and adjacent to hydraulic structures where aeration and energy dissipation are fundamental. Three representative cases are investigated: an impinging jet, a headcut jet, and a spillway chute with stilling basin.

This assessment is of utmost importance for hydraulic engineering researchers and practitioners, because the Volume-of-Fluid method (VoF) is still frequently applied in flows with significant air-water mixing, despite tending to null aeration as mesh resolution increases, which leads to unrealistic results.

In contrast, the Complete Two-Phase Euler model - despite still being considered computationally demanding and sensitive to numerical instability - addresses several limitations of the VoF approach and offers a more reliable cost-benefit performance. It is particularly suitable for simulating high-velocity, high-turbulence, and intensely aerated conditions typical of many hydraulic structures.

This work aims to promote the adoption of the Complete Two-Phase Euler model in the hydraulic engineering community. Three case studies are presented and made publicly available through an open repository, contributing to transparency, reproducibility, and further development.

Introduction

The National Laboratory of Civil Engineering of Portugal (LNEC) has over 74 years of experience in the validation and optimisation of hydraulic structures. Historically, physical modelling has been the main tool of these studies, despite inherent limitations such as scale effects, experimental constraints, and the substantial costs associated with model construction and operation.

Over the last 15 years, LNEC has progressively incorporated Computational Fluid Dynamics (CFD) into its workflow. This integration has enabled the rapid evaluation of multiple design alternatives at a reduced cost, while also providing access to full-domain flow variables that are typically difficult or impossible to capture in physical experiments.

In 2024, LNEC proposed a six-step integration methodology that strategically combines physical and numerical modelling. This approach leverages the strengths of each technique while mitigating their individual limitations, and has proven to be the most effective strategy for hydraulic structure studies [1].

Nonetheless, CFD simulations in hydraulic engineering present significant challenges. The large-scale and geometrically complex nature of hydraulic structures demands extensive computational domains capable of resolving a wide range of temporal and spatial scales. Even with access to high-performance computing (HPC) infrastructure, achieving sufficiently refined mesh resolution across the entire domain remains difficult. This adds complexity to the accurate modelling of turbulence and air–water interaction [2].

Methodology

This study evaluates the performance of the Complete Two-Phase Euler–Euler model, implemented in the twoPhaseEulerFoam solver, as an alternative to the more commonly used Volume-of-Fluid (VoF) approach [3,4,5]. While VoF is simpler to apply, it tends to suppress aeration effects with increased mesh refinement, which can lead to unrealistic results in highly aerated flows. The Euler–Euler approach addresses many of these shortcomings, offering a more physically accurate treatment of air–water mixtures. However, it is often considered more complex, computationally demanding, and susceptible to numerical instabilities [6].

The primary objective of this work is to provide practical guidelines for researchers and practitioners in hydraulic engineering aiming to model flows characterised by high velocities, intense turbulence, and strong aeration.

The model is applied to three distinct case studies, each representative of frequently encountered flow conditions in hydraulic structures:

- Impinging jet into a plunge pool
- Headcut jet
- Spillway chute stilling basin

All case studies are characterised by highly turbulent and aerated flows. A comprehensive assessment is carried out, focusing on key aspects such as accuracy, numerical convergence, stability, usability, computational cost, and overall efficiency. Multiple mesh resolutions are tested, and different turbulence models are compared, including *mixture $k-\epsilon$* , and *$k-\omega$ SST*.

Conclusions

The Two-Phase Euler model is a crucial tool for modelling aerated flows in hydraulic engineering. It is accurate at intermediate mesh resolutions, showing an acceptable computational effort and without significant numerical instabilities. Various aeration effects are modelled, such as compressibility and flow bulking.

Therefore, the twoPhaseEulerFoam solver has an efficient cost–benefit relation and plays an important role in modelling hydraulic structure flows in the presence of high velocity, high turbulence levels, and intense aeration.

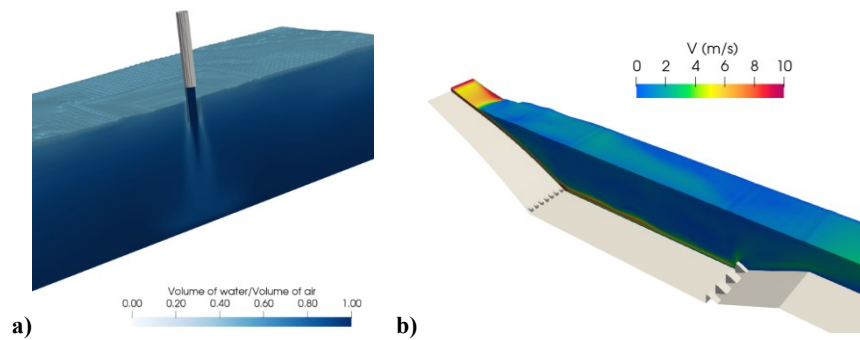


Figure 1: a) Impinging jet; b) Hydraulic-jump dissipation basin

Acknowledgements

The authors thank all those involved in the organisation of the 7th FOAM@Iberia meeting and to all the contributors that will enrich this event.

References

- [1] L. S. Mendes, A. Muralha, J. F. Melo, T. Viseu, “Why computational fluid dynamics is mandatory in physical modelling of large hydraulic structures: a six-step methodology of integration”, 8th IAHR Europe Congress, Lisbon – Portugal, 4-7 June, 2024.
- [2] L. S. Mendes, “Computational Fluid Dynamics of Aerated Flows in Spillways Chutes,” PhD thesis, Universidad de Cantabria, Santander, España, 2022. <http://hdl.handle.net/10902/23909>
- [3] L. S. Mendes, J. L. Lara, and M. T. Viseu, “Is the Volume-of-Fluid Method Coupled with a Sub-Grid Bubble Equation Efficient for Simulating Local and Continuum Aeration?” *Water*, vol. 13, no. 11, p. 1535, 2021. <https://doi.org/10.3390/w13111535>
- [4] L. S. Mendes, J. L. Lara, and T. Viseu, “Do the Volume-of-Fluid and the Two-Phase Euler Compete for Modeling a Spillway Aerator?” *Water*, vol. 13, no. 21, p. 3092, 2021. <https://doi.org/10.3390/w13213092>
- [5] A. Muralha, J. F. Melo, and H. M. Ramos, “Validation of Computational Methods for Free-Water Jet Diffusion and Pressure Dynamics in a Plunge Pool”, *Applied Sciences*, vol. 15, no. 4, p. 1963, 2025. <https://doi.org/10.3390/app15041963>
- [6] A. Muralha, J. F. Melo, and H. M. Ramos, “Assessment of CFD Solvers and Turbulent Models for Water Free Jets in Spillways,” *Fluids*, vol. 5, no. 3, p. 104, 2020. <https://doi.org/10.3390/fluids5030104>