A general overview of OpenFOAM as an open source numerical wave tank

Devolder Brecht[1], Troch Peter[2] and Rauwoens Pieter[3]

[1] Ghent University and KU Leuven (Construction Technology Cluster, Campus Bruges), Department of Civil Engineering, Technologiepark 904, 9052 Ghent, Belgium
E-mail: Brecht.Devolder@UGent.be
[2] Ghent University, Department of Civil Engineering, Technologiepark 904, 9052 Ghent, Belgium
[3] KU Leuven, Department of Civil Engineering, Construction Technology Cluster, Campus Bruges, Spoorwegstraat 12, 8200 Bruges, Belgium

During the last years, the complexity of coastal and offshore structures has grown significantly. One example is the evolution of foundation types for offshore wind turbines from basic monopiles towards complex jacket structures and very recently up to innovative floating wind turbines. This increasing complexity results in a great interest for numerical tools in addition to analytical verification methods and traditional small-scale experimental modelling. Numerical modelling is nowadays very popular for R&D activities due to the availability of extremely powerful computational resources. Since a few years, the coastal and offshore community shows interest in numerical wave tanks (NWTs) to answer specific design questions for a structure installed offshore or nearshore.

A NWT is the equivalent of a physical wave basin and they share the same objective: reproducing the physics as observed in the ocean or the sea in a controllable environment to study physical processes in detail or to check design criteria. In this overview, the NWT is implemented in the open source Computational Fluid Dynamics (CFD) software OpenFOAM. Over the past years, three main branches are developed and made available by the OpenFOAM Foundation, OpenCFD and the foam-extend community. OpenFOAM is used (i) to solve the Navier-Stokes equations to calculate the hydrodynamics (i.e. pressure and velocity) and (ii) to apply the Volume of Fluid method to determine the position of the free water surface in the numerical wave flume. Furthermore, boundary conditions for wave generation and absorption are mostly adopted from toolboxes such as waves2foam, IHFOAM and olaFlow. In addition, those three toolboxes are able to simulate the flow through porous media such as breakwaters for example. Also currents, with or without waves, are generated in NWTs in order to enhance the reproduction of the physics observed in reality. Recommendations for turbulence modelling (i.e. RANS or LES) are provided by many authors. We emphasise our recent work where we enhanced the predictive skills of NWTs by applying buoyancy-modified RANS turbulence models [1,2]. Furthermore, a NWT is very suitable to study fluid-structure interactions such as wave run-up on structures, wave attack on storm walls and floating structures subjected to specified wave conditions. As an example, we developed an accelerated coupling algorithm for simulations of floating structures in a NWT [3]. Lastly, NWTs are coupled with a sediment transport module to study the evolution of the bottom in the swash zone under wave and/or current action [4].

Nowadays, NWTs are under continuous development to reduce the disadvantages. Firstly, enhanced predictions of complex processes such as the hydraulic stability of rubble mound breakwaters are still lacking. Secondly, NWTs suffer from long calculation times especially for long duration tests. Thirdly, the obtained results are only as good as the physics involved in the equations being solved. Therefore, physical wave basis are always needed complementary to NWTs such as the coastal and offshore basin which is now under construction at the GreenBridge science park in Ostend (Belgium).

The first author is Ph.D. fellow of the Research Foundation – Flanders (FWO), Belgium (Ph.D. fellowship 1133817N).

References

[1] Devolder et al., 2017. Application of a buoyancy-modified *k-ω SST* turbulence model to simulate wave run-up around a monopile subjected to regular waves using OpenFOAM®. *Coastal Engineering, 125*, 81–94.
[2] Devolder et al., 2018. Performance of a buoyancy-modified *k-ω* and *k-ω SST* turbulence model for simulating wave breaking under regular waves using OpenFOAM®. Under review for Coastal Engineering.
[3] Devolder et al., 2017. Accelerated numerical simulations of a heaving floating body by coupling a motion solver with a two-phase fluid solver. *Book of abstracts of the 7th International Conference on Advanced Computational Methods in Engineering*.

[4] Vandebeek et al., 2017. Towards a climate resilient coast: Numerical modeling of wave overtopping and wave loads considering the influence of sediment transport. *Book of abstracts of the VLIZ Marine Science Day*.

Keywords: numerical modelling; CFD; OpenFOAM; numerical wave tank