Numerical modelling of wave interaction with coastal and offshore structures using a CFD solver

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

[1] Department of Civil Engineering, Ghent University, Technologiepark 904, 9052 Zwijnaarde (Ghent), Belgium
E-mail: Brecht.Devolder@UGent.be

[2] Construction TC, Technology Campus Ostend, Department of Civil Engineering, KU Leuven, Zeedijk 101, 8400 Oostende, Belgium

Sustainable development of coastal and offshore structures – from conceptual layouts, optimisation, stability and environmental assessment to detailed design – requires a thorough understanding of coastal and offshore processes. Numerical modelling is a suitable tool to investigate these physical processes and to deliver valid, accurate, efficient and usable results and solutions. Parallel to the increase of the available computational power, the use of numerical models is growing rapidly, also in the design of all kinds of coastal and offshore structures, such as breakwaters, wave energy converters, etc.

In this work, the Computational Fluid Dynamics (CFD) software OpenFOAM (http://openfoam.org/) has been used to study wave interaction with coastal and offshore structures. The numerical simulations are performed in a numerical wave tank, which is the equivalent of a physical wave tank used for experimental model testing at a smaller scale. However, the idea behind both wave tanks is the same: reproducing the physics as observed in the ocean or the sea in a controllable environment to study particular physical processes in detail. The numerical wave tank is characterised by a length, width and height. Boundary conditions are needed at the sides of the tank, for example to generate and absorb waves. The CFD software solves a set of partial differential equations, the Navier-Stokes equations, representing the physics with a very high accuracy. Therefore, the numerical wave tank is split up in a large amount of small grid cells and the governing differential equations are discretised to algebraic equations using a finite volume method. In each grid cell, the algebraic equations are solved numerically using linear systems, taking the boundary conditions into account.

In this study, we present several simulations of coastal and offshore structures using a CFD solver:
- Wave propagation in a two-dimensional wave flume,
- Wave breaking on a sloping beach,
- Wave run-up on a monopile,
- Modelling of wave energy converter arrays.

The first author is Ph.D. fellow of the Research Foundation – Flanders (FWO), Belgium (Ph.D. fellowship 1133815N).

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