Investigation on the Use of a Multiphase Eulerian CFD solver to simulate breaking waves - DTU Orbit (08/11/2017)

Investigation on the Use of a Multiphase Eulerian CFD solver to simulate breaking waves

The main challenge in CFD multiphase simulations of breaking waves is the wide range of interfacial length scales occurring in the flow: from the free surface measurable in meters down to the entrapped air bubbles with size of a fraction of a millimeter. This paper presents a preliminary investigation on a CFD model capable of handling this problem. The model is based on a solver, available in the open-source CFD toolkit OpenFOAM, which combines the Eulerian multi-fluid approach for dispersed flows with a numerical interface sharpening method. The solver, enhanced with additional formulations for mass and momentum transfer among phases, was satisfactorily tested against an experimental bubble column flow. The model was then used to simulate the propagation of a laboratory solitary breaking wave. The motion of the free surface was successfully reproduced up to the breaking point. Further implementations are needed to simulate the air entrainment phenomenon

General information

State: Published
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Coastal and Maritime Engineering
Authors: Tomaselli, P. D. (Intern), Christensen, E. D. (Intern)
Number of pages: 10
Publication date: 2015

Host publication information

Title of host publication: Proceedings of the ASME 34th International Conference on Ocean, Offshore and Arctic Engineering
Article number: OMAE2015-41640
BFI conference series: International Conference on Ocean, Offshore and Arctic Engineering (5010067)
Main Research Area: Technical/natural sciences
Conference: ASME 2015 34th International Conference on Ocean, Offshore and Arctic Engineering, St John's, Newfoundland, Canada, 31/05/2015 - 31/05/2015
Publication: Research - peer-review › Article in proceedings – Annual report year: 2015