International Journal of Offshore and Polar Engineering (ISSN 1053-5381)

Vol. 21, No. 3, September 2011, pp. 186–191

http://www.isope.org/publications

Two-Phase-Flow Unstructured Grid Solver: Application to Tsunami Wave Impact

Alioune Nar Sambe
Laboratoire de Sondages Electromagnétiques de l'Environnement Terrestre et
Institut de Mathématiques de Toulon, Université du Sud Toulon Var, La Garde, France

Frédéric Golay
Institut de Mathématiques de Toulon, Université du Sud Toulon Var, La Garde, France

Damien Sous, Philippe Fraunié and Vincent Rey
Laboratoire de Sondages Electromagnétiques de l'Environnement Terrestre
Université du Sud Toulon Var, Centre National de la Recherche Scientifique, La Garde, France

Richard Marcer and Christine De Jouette
Principia R&D, La Ciotat, France

This paper describes the CFD of incompressible air-water flows by solving Euler equations with a 2-phase flow model leading to a hyperbolic system of conservation laws solved with a finite volume discretization on unstructured grids. An artificial compressibility approach allows for a fully explicit scheme for an efficient parallel implementation. The numerical model is based on a low Mach number preconditioning and a 2nd-order Riemann solver. Applications to breaking waves on a 15% slope with and without the influence of macro-roughness are performed concerning impact and run-up processes on the beach.

INTRODUCTION

Great improvements have been brought to the knowledge of the hydrodynamics and the general processes occurring in the surf zone, which is widely affected by breaking waves (Peregrine, 1983; Christensen et al., 2006).

Indeed, breaking waves play a very important role in marine hydrodynamics, both for ocean physics and engineering applications. Breaking is the most dominant phenomenon in the surface-wave energy budget. In the naval hydrodynamics context, as for coastal dynamics, a reliable and fast model is necessary to perform parametric investigation so as to optimize sea defence systems. In particular, the kinematics of impact and run-up on structures and beaches are drastically important in the case of hazardous events such as surges and tsunamis.

Concerning wave-breaking modeling, the classical, full Navier-Stokes solvers are time-consuming (e.g. Vincent et al., 2004; Guignard et al., 2001; Biausser et al., 2004), while fast models based on Boussinesq equations are unable to compute wave breaking (Grilli et al., 1989, 2001; Guyenne and Grilli, 2006). Here we present a new, fast 3D 2-phase flow solver, based on a finite volume discretization on unstructured grids with subdomain decomposition allowing for an efficient parallel implementation. This model has already been validated in the Yasuda et al. (1997) experimental and numerical test case with convincing numerical performances and computational time (Helluy et al., 2005).

Here, we use a 2-phase flow model where equations of conservation are solved in a single domain by introducing a state equation depending on the volume fraction of water and air (Chanteperdrix et al., 2002; Saurel and Abgrall, 1999). This volume fraction is transported without any interface tracking. A low Mach number assumption applied to Euler equations leads to a hyperbolic system of conservation laws solved by appropriate fast numerical methods (Helluy et al., 2005).

More recently, some authors have used a similar approach (Dias et al., 2010) and performed comparisons with the classical, full Navier-Stokes model (Braconnier et al., 2009). Our model should be considered a compromise between computational cost and relevance of physical modeling. Indeed, as in the Boussinesq approach, turbulence and viscosity are neglected, which avoids the discretization of the Laplacian operator and allows for a fast hyperbolic solver, hence 3D computations. The equation of state (EOS) of our average mixed model is artificial but numerically relevant, and we do not need to build the interface as in the VOF method (Kleefsman et al., 2005). Such an approach limits computational time and allows for 3D simulations, which require up to now unusual computers for both SPH methods (e.g. Khayyer et al., 2009) and the classical bi-fluid approach (e.g. the work of Fuster et al., 2009, which uses the Gerris code with mesh refinement by octree).

The present study aims to analyse, by numerical modeling, the influence of macro-roughness on wave-breaking and run-up processes. By contrast to skin friction, which only acts on the bottom shear stress, the presence of macro-roughness produces modifications of the flow field, i.e. accelerations and decelerations, up to the free surface. Our primary objective is thus to understand how the roughness elements will affect the breaking dynamics, location of the breaking point, energy dissipation and run-up height. In this context, the inviscid numerical model appears to be a relevant tool.

MATHEMATICAL MODEL

We use an Eulerian 2-phase flow model leading to a hyperbolic system of conservation laws for mass and momentum. The

Received December 14, 2009; revised manuscript received by the editors December 16, 2010. The original version (prior to the final revised manuscript) was presented at the 19th International Offshore and Polar Engineering Conference (ISOPE-2009), Osaka, June 21–26, 2009.

KEY WORDS: Wave breaking, finite volume, Godunov scheme, air-water interface, run-up, macro-roughness.