Ocean Engineering 106 (2015) 298–316

Contents lists available at ScienceDirect

Ocean Engineering

journal homepage: www.elsevier.com/locate/oceaneng

Numerical simulations of 2-D steady and unsteady breaking waves

Guido Lupieri *, Giorgio Contento [1]

Department of Engineering and Architecture, University of Trieste, via Valerio, 10, 34127 Trieste, Italy

ARTICLE INFO

Article history:
Received 3 July 2014
Accepted 5 July 2015
Available online 29 July 2015

Keywords:
Breaking waves
Steady and unsteady breakers
Turbulence models
OpenFOAM

ABSTRACT

In this work we analyze by means of numerical simulations the features of breaking of two dimensional free surface waves induced by a body or a sloping bottom. The sample cases selected for the simulations characterize different aspects of wave breaking, thus they are supposed to represent rather widely a problem of large interest for ship hydrodynamics and ocean engineering applications. The simulations considered are: wave breaking induced by a fully submerged hydrofoil towed in calm water at constant speed; shallow water waves breaking on a sloping beach in spilling and plunging mode; regular intermediate depth waves breaking gently over a weakly submerged horizontal circular cylinder at a low Keulegan–Carpenter number. Each simulated case is supported by detailed comparisons with experimental data in time and frequency domain. The results presented have been obtained adopting a standard RANS approach. They show a generally good reproduction of the wave breaking characteristics even though it is rather clear that there is a case dependent potential loss of accuracy in the presence of pronounced foamy flow.

1. Introduction

Wave breaking plays an important role in ship/marine hydrodynamics and in offshore/coastal engineering as it relates, among others, to wave loads on floating or fixed bodies, to energy loss of wind waves and to ship resistance in calm water or in a seaway. Wave breaking is also related to the ultimate behavior of steep deep water waves under modulational (Benjamin–Feir) instability. In this respect, the scientific community involved in free surface hydrodynamics has fed both laboratory measurements and numerical studies. RANS models still represent the bulk of the numerical simulations. The understanding of the actual effect of turbulence in the two-phase flow resulting from breaking is still a challenging problem, both in experimental measurements and in numerical simulations. For instance, simplified models have been developed and applied to overcome the difficulties in handling the typical unsteady foamy flow of the breakers. Muscari and Di Mascio (2004) have proposed a wave breaking model parameterization useful to account for average energy loss in the RANS computations of ship resistance in calm water.

The simulation of two-dimensional breaking waves is often considered as a common starting point for understanding, handling or calibrating a new solver for the dynamics of interface problems with breaking. A common technique for modeling 2-D flows in turbulent regime is RANS approach. In this case, there are at least three relevant aspects to take care of: (a) the parameterization of the flow with turbulence models acts as an intrinsic cutoff of the high frequency space/time fluctuations and therefore it allows to capture only some characteristics of the flow (Iaccarino et al., 2003); (b) turbulence modeling in 2-D is always a thorny charge because of a 3-D intrinsic nature of the phenomenon (Lilly, 1969), and even if the most popular RANS turbulence models, from Spalart–Allmaras to k–ϵ, have been extensively calibrated and adopted also in 2-D flows (Menter, 1994), the whole energy cascade mechanism is not straightly accounted for and this aspect could still reveal further weaknesses in the solution (Zhao et al., 2004), (c) since turbulence is a multi-scale phenomenon, a range of lengths (for instance the sizes of the vortices eventually present in the flow) and frequencies is present but, when designing a RANS simulation, it is a request to identify a unique reference value, for instance, for specific dissipation or turbulent kinetic energy (namely the amount of fluctuations in the velocity field) (Zhao and Armfield, 2010). Moreover most of the standard experiments in the marine hydrodynamics field are conducted at relatively low Reynolds numbers, typically of order of 10^5–10^6. In these cases, the adoption of RANS methods can lead to oversmoothed free surface profiles, moving the position of the crests/hollows, inhibiting the entrapment of air bubbles and removing implicitly high frequency terms in the flow field and in the free surface elevation.

Starting from the considerations above, the goal of this work is to reproduce some of the main characteristics of complex two-phase

* Corresponding author. Tel.: +39 040 5582955.
E-mail addresses: glupieri@units.it (G. Lupieri), contento@units.it (G. Contento).
[1] Tel.: +39 040 5583424.

http://dx.doi.org/10.1016/j.oceaneng.2015.07.014
0029-8018/

flows with breaking, namely free surface elevation, its macro-time scales and pressure field, adopting a standard RANS approach. Solid experimental data are used as reference. The numerical experiments are executed with and without modeling of turbulence, in order to evince, when time and space resolutions are adequate, useful indications on the same targets.

Consistently with the RANS approach, the main instruments for the analysis are averages (over a number of periods and in phase), reproducing here the free surface in statistically meaningful terms. This is done in all cases analyzed. Furthermore, in order to clarify some aspect of the phenomena or to refer to solid experimental and numerical results from other authors, time series at fixed gauges and snapshots will be presented too.

The selected numerical simulations are:

Case I : steady and unsteady breaking induced by a fully submerged hydrofoil at constant speed in calm water;
Case II : ultra shallow water cnoidal waves breaking in spilling and plunging mode on a sloping beach (ramp);
Case III : breaking of intermediate water depth regular waves, induced by a weakly submerged horizontal circular cylinder at a low Keulegan–Carpenter number.

The first two simulations (Case I and Case II) are supported by the results of celebrated experiments, Duncan (1981, 1983, 2001), De Blasi et al. (2000) and Ting and Kirby (1994, 1995, 1996) respectively, whereas Case III has been previously studied in the hydrodynamic laboratory of the University of Trieste by one of the authors (Contento and Codiglia, 2001). The cases selected characterize different aspects of wave breaking induced by a solid boundary/body, thus they are supposed to represent a rather wide variety on a problem of interest for engineering applications.

In the first case, a submerged 2D hydrofoil travels in steady incident flow, generating a wave train. Depending on the Froude number, on the foil submergence and on the angle of attach, the wave train may ultimately break. The incident flow is originally in a laminar regime but breaking makes it locally turbulent. Furthermore the breaker has been observed to pulsate back-and-forth with a well-defined periodicity that depends on the Froude number (Duncan, 1981, 1983, 2001).

In the second case, there is no body inducing breaking. Cnoidal waves, generated by a wavemaking boundary, travel and break on a sloping ramp. Depending on the characteristics of the incident wave, two different types of breaking events may take place, spilling and plunging breakers (Ting and Kirby, 1994, 1995, 1996).

In the third case, a deep water regular wave train generated by a wavemaking boundary, breaks gently on a weakly submerged circular cylinder. Furthermore, the specific case examined is characterized by a low Keulegan Carpenter number and the wave–body interaction leads to a steady streaming around the cylinder surface that induces a pressure field playing a crucial role in the surface elevation as a suction effect on the wave throats and breaking. This has been observed experimentally by Contento and Codiglia (2001).

The paper is organized as follows:

- in Sections 2 and 3 the mathematical method and the numerical approach are described briefly;
- in Sections 4–6 each physical problem is initially outlined with references to previous studies, then providing details on the computational set up; finally the specific results of interest are shown with comments.

The tool used for this investigation is the OpenFOAM (2012) library. The k–ωSST of Menter (1994) has been considered for turbulence modeling. In this finite volume library, the solver for the problems enounced is *interFoam* that includes, as standard for the treatment of free surface fluxes, the volume of fluid technique of Hirt and Nichols (1981). In this work the library has been enriched by a numerical wave absorber designed according to Clement (1996), Smith (2009), and Wang et al. (2007) and added to avoid undesired reflections of waves from the boundaries. A wavemaking boundary has been implemented as well.

2. Mathematical model

The governing equations for incompressible Newtonian fluid are the momentum and the mass conservation equation:

$$\frac{\partial(\rho u_i)}{\partial t}+\frac{\partial(\rho u_j u_i)}{\partial x_j}=-\frac{\partial p}{\partial x_i}+\frac{\partial}{\partial x_j}\left(\mu\left(\frac{\partial u_i}{\partial x_j}+\frac{\partial u_j}{\partial x_i}\right)\right) \tag{1}$$

$$\frac{\partial u_i}{\partial x_i}=0 \tag{2}$$

where ρ is the fluid density, u_i is the velocity component, p is pressure without hydrostatic term, μ is the dynamic viscosity, t and x_i the time/space independent variables.

Broadly speaking, for large Reynolds numbers the Navier–Stokes equations can be reformulated in terms of Reynolds averages. Then, to achieve the closure of the new set of equations, additional equations are added in order to redefine the eddy viscosity. In this paper, the turbulence model in use is the k–ω Shear Stress Transport (Menter, 1994) that consists of two extra transport equations, for the turbulent kinetic energy k and for the specific turbulent dissipation ω respectively. Being interested in two phase flows (a coupled air–water interface system), it is possible to deal with interface capturing methods such as the VOF technique of Hirt and Nichols (1981). The idea is to use a scalar function α to represent the phase of the fluid in each cell, therefore for the viscosity μ and the density ρ in Navier–Stokes equations we have

$$\begin{cases}\mu=\mu_{water}\alpha+\mu_{air}(1-\alpha)\\ \rho=\rho_{water}\alpha+\rho_{air}(1-\alpha)\end{cases} \tag{3}$$

For the scalar function α the following equation holds:

$$\frac{\partial\alpha}{\partial t}+\frac{\partial(u_i\alpha)}{\partial x_i}=0 \tag{4}$$

The function α is bounded between 1 (if only water is present in a control volume) and 0 (if only air is present) over an extremely small layer. This can lead to numerical difficulties associated with the discretization of the convection term in Eq. (4). This in turn results in smearing of the interface. Following Rusche (2002) and Maki (2011), we have used a modified transport equation with an additional convective term that serves to keep the interface sharp:

$$\frac{\partial\alpha}{\partial t}+\frac{\partial(u_i\alpha)}{\partial x_i}+\frac{\partial(w_i\alpha)}{\partial x_i}=0 \tag{5}$$

where w_i is an artificial velocity field that is directed normal to and towards the interface. The relative magnitude of the artificial velocity is determined with the following expression:

$$w_i=K_c n_i^* \max\frac{|n_i^* F_l|}{|S_i|} \tag{6}$$

where K_c is an adjustable coefficient that determines the magnitude of the compression, n_i^* is the interface unit normal vector, F_l is the flux and S_i is the surface area vector.

These equations complete the mathematical formulation of the two phase flow model. In the following, unless differently specified, the nominal free surface elevation (air–water interface) is referred to $\alpha=0.5$. It has been shown in the literature (Chen et al., 2014; Maki, 2011) that Eq. (5) allows mass conservation at a very