



Two-phase flow modelling of spilling and plunging breaking waves

Zhihua Xie*

Centre for Computational Fluid Dynamics, University of Leeds, Leeds LS2 9JT, UK

ARTICLE INFO

Article history:

Received 8 November 2011
Received in revised form 6 June 2012
Accepted 26 July 2012
Available online 7 September 2012

Keywords:

Breaking waves
Two-phase flow model
Surf zone dynamics
Reynolds-averaged Navier–Stokes (RANS) equations
VOF method
Turbulence modelling

ABSTRACT

A two-phase flow model, which solves the flow in the air and water simultaneously, has been employed to investigate both spilling and plunging breakers in the surf zone with a focus during wave breaking. The model is based on the Reynolds-averaged Navier–Stokes equations with the $k-\epsilon$ turbulence model. The governing equations are solved using the finite volume method, with the partial cell treatment being implemented in a staggered Cartesian grid to deal with complex geometries. The PISO algorithm is utilised for the pressure–velocity coupling and the air–water interface is modelled by the interface capturing method via a high-resolution volume of fluid scheme. Numerical results are compared with experimental measurements and other numerical studies in terms of water surface elevations, mean flow and turbulence intensity, in which satisfactory agreement is obtained. In addition, water surface profiles, velocity and vorticity fields during wave breaking are also presented and discussed. It is shown that the present model is capable of simulating the wave overturning, air entrainment and splash-up processes.

© 2012 Elsevier Inc. All rights reserved.

1. Introduction

Wave breaking plays an important role in air–sea interaction, surf zone dynamics, nearshore sediment transport, marine hydrodynamics, and wave–structure interaction. Over the last three decades, significant advances have been made in the theoretical, experimental, and numerical studies of the characteristics of breaking waves (see [1–4] for reviews). However, little attention has been paid to investigate breaking waves during the wave overturning process. Wave breaking is responsible for the wave energy dissipation and the generation of turbulence, vorticity and nearshore currents in the surf zone.

Much of our knowledge of breaking waves comes from laboratory measurements. Several systematic studies have been carried out for steady breaking waves [5], quasi-steady breaking waves [6], unsteady deep-water breaking waves [7–11], and breaking waves in the laboratory surf zone [12–17]. Overall, with the development of measurement techniques, physical experiments have provided much insight into the kinematics and dynamics of breaking waves.

Many numerical models have been developed to simulate periodic breaking waves in the surf zone. One of these is the Boussinesq-type model [18–20], which is widely used in the nearshore wave modelling. With developments of CFD (computational fluid dynamics) and increases in computer power, recent models for studying free surface flows, including breaking waves, solve the Navier–Stokes equations coupled with a free surface calculation (see [21] for comprehensive modelling applications and methodologies for water waves). Lemos [22] presented an early study for breaking solitary waves and periodic breaking waves using the RANS (Reynolds-averaged Navier–Stokes) model, which was based on the SOLA-VOF code [23] with the standard $k-\epsilon$ turbulence model. Takikawa et al. [24] investigated a plunging breaker on a slope. The RANS equations were solved by a modified SMAC (Simplified Marker and Cell) method [25] and the calculation was initialised from another simulation result to model the wave breaking process. A useful numerical model to study breaking waves in the surf

* Present address: Hydro-environmental Research Centre, School of Engineering, Cardiff University, Cardiff CF24 3AA, UK. Tel.: +44 (0) 29 2087 6814; fax: +44 (0) 29 2087 4939.

E-mail address: zhihua.xie@hotmail.com

zone was the COBRAS (COrnell BRaking waves And Structures) model developed by Lin and Liu [26,27], who combined the modification of the RIPPLE code [28] with implementation of the algebraic Reynolds stress $k-\epsilon$ turbulence model. The RANS equations were solved by the two-step projection method [29] in the finite difference form and the VOF (volume-of-fluid) method was employed to capture the water surface. Periodic breaking waves on a sloping beach were investigated and compared with the experimental measurements [13–15]. Good agreement between numerical results and experimental data was obtained in terms of water surface profiles, mean velocities, and turbulent kinetic energy. The plunging jet of an overturning wave on constant water depth was computed in [27]. Since then, several investigations have been performed to study breaking waves in the experiments [13–15]. Bradford [30] utilised the commercial software FLOW-3D to investigate spilling and plunging breakers in the surf zone and compared with the experimental data. A comparison of the numerical results obtained by different turbulence models was made, and detailed water surface elevation, time averaged fields for the velocities and turbulence were analysed. It was found that the location of initial wave breaking was sensitive to the representation of waves from the inlet boundary. In addition, the model had difficulty in capturing the plunging jet, which may be attributed to a lack of spatial resolution or the VOF scheme used in the simulation. Comparison between different turbulence models indicated that a one equation turbulence model was inadequate to get good results, and the RNG (Re-Normalized Group) turbulence model predicted lower turbulence intensities in the outer surf zone compared to the $k-\epsilon$ model. Mayer and Madsen [31] investigated spilling breakers [15] by solving the RANS equations with the $k-\omega$ turbulence model. A surface tracking approach and a VOF method were employed to model the interface and it was shown that the VOF method gave better results. Zhao et al. [32] performed a numerical simulation of spilling and plunging breakers. The space filtered Navier–Stokes equations with a multi-scale turbulence model were proposed and solved by the finite difference approach. Improved agreement with experimental measurements was obtained in terms of water surface elevations, wave height distribution and mean velocities when compared to the RANS models [26,30]. In addition, it was found that turbulent production was mainly located at the front of the wave whereas turbulent dissipation was mainly located at the rear face of the wave. Similar to Bradford [30], the plunging jet was not captured in the plunging breaker case because the air entrainment was not taken into account as indicated by Zhao et al. [32], but the plunging jet of an overturning wave was presented in that paper. Shao [33] presented the simulation of both spilling and plunging breaking waves by the SPH (smoothed particle hydrodynamics) method coupled with the $k-\epsilon$ model and later extended with the LES Smagorinsky model [34]. The curling forward of the plunging jet was captured by the SPH method and it was shown that the SPH method provided a useful tool to investigate surf zone dynamics. Hsieh et al. [35] solved the RANS model with the VOF and embedding methods to simulate spilling breakers. More recently, Bakhtyar et al. [36] employed the RANS equations with the standard $k-\epsilon$ turbulence model to investigate the turbulent flow in the surf and swash zones.

It is worth remarking that all the models discussed above are based on one-phase flow, in which only the flow in the water is considered in the computation. In order to take the air into account for wave breaking, several two-phase flow models have been developed, in which both flows in the air and the water are solved. Hieu et al. [37] developed a two-phase flow model to investigate two-dimensional (2D) breaking waves in the surf zone. The sub-grid scale Smagorinsky model, which is similar to the large eddy simulation, was employed to get the turbulent eddy viscosity. Spilling breaking waves were considered in that paper and compared with the experiment [15]. Wang et al. [38] performed a 2D study of spilling breaking waves in the surf zone without turbulence modelling. A mass conservative level set method was used for capturing the air–water interface and the solver was based on a curvilinear coordinate system. There are also some attempts for three-dimensional (3D) large eddy simulation of breaking waves using one-phase [39,40] and two-phase [41,42] flow models, and Christensen [43] presented the 3D simulation of spilling and plunging breakers in the surf zone.

It is worth mentioning that as the details of the water surface elevation, mean flow and turbulence fields for both spilling and plunging breakers were presented in [13–15], these laboratory studies have been considered by many researchers to test their models for simulating breaking waves and turbulence in the surf zone. However in most studies, the detailed information during wave breaking, such as the plunging jet and air entrainment, were not captured in the simulation. Thus, in this study, a recently developed two-phase flow model [44], which is different from Hieu et al. [37] and Wang et al. [38] in numerical method and turbulence modelling approach, is employed to investigate both spilling and plunging breakers in the surf zone with a focus during wave breaking.

The organisation of this paper is as follows. The description of the mathematical model for the two-phase flow is described in Section 2. The numerical method is presented in Section 3. Both spilling and plunging breakers in the surf zone are simulated in Section 4. Numerical results are compared with available experimental data and other numerical studies. Detailed information during wave breaking is presented and discussed. Finally, conclusions are drawn in Section 5.

2. Mathematical model

2.1. Governing equations

The governing equations for incompressible Newtonian fluid flow are the Reynolds-averaged Navier–Stokes equations. Mass conservation is described by the continuity equation

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \mathbf{u}) = 0, \quad (1)$$

Download English Version:

<https://daneshyari.com/en/article/1704644>

Download Persian Version:

<https://daneshyari.com/article/1704644>

[Daneshyari.com](https://daneshyari.com)