Journal of Fluids and Structures 32 (2012) 121–134

Contents lists available at SciVerse ScienceDirect

Journal of Fluids and Structures

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

Impulsive plunging wave breaking downstream of a bump in a shallow water flume—Part II: Numerical simulations

Bonguk Koo, Zhaoyuan Wang, Jianming Yang, Frederick Stern*

IIHR-Hydroscience & Engineering, University of Iowa, Iowa City, IA 52242-1585, USA

ARTICLE INFO

Article history:
Received 15 November 2010
Accepted 22 October 2011
Available online 1 December 2011

Keywords:
Plunging wave breaking
Impulsive
Wave–body interaction
Air entrainment
Volume-of-fluid

ABSTRACT

Part II of this two-part paper presents the simulation results of the plunging wave-breaking generated by impulsive flow over a submerged bump fixed in a shallow water flume using the exact experimental initial and boundary conditions provided in Part I of this study. The overall plunging wave breaking process is described with major wave breaking events identified: jet plunge, oblique splash and vertical jet. These major events repeat up to four times before entering the chaotic breaking phase. The plunging wave breaking process of the simulations shows a similar time line as the experiments consisting of startup, steep wave formation, plunging wave, and chaotic wave breaking swept downstream time phases. Wave breaking profile, air entrainment, velocity, vorticity, flume bottom pressure, and energy are analyzed and compared with the experimental results. The simulations qualitatively predict all four time phases, all four plunging events and their sub-events. The wave profile and location at the maximum height is very close to the experiment results. The flume flow and velocity demonstrate the same flow trend as the experiments but with reduced velocity magnitudes. The simulations show similar bottom pressure to the experiments but with large oscillations, and the post-breaking water elevations are larger as compared to the experimental results.

1. Introduction

Plunging wave breaking is one of the most violent phenomena of air–water interface interactions, producing strong turbulence with large amounts of air bubbles, water droplets, jets, and sprays. These phenomena commonly occur in ship flows and are one of the main sources of the underwater sounds and white-water wakes, which are of great importance for signature of ships. Many experimental and computational studies for the plunging wave breaking have been done in the past few decades. Early experimental studies are focused on wave geometric properties (Bonmarin, 1989), wave breaking process (Tallent et al., 1990), energy dissipation (Melville, 1994), jet characteristics and air entrainment (Chanson and Fang, 1997), and turbulence (Chang and Liu, 1999). With the Particle Image Velocimetry (PIV) techniques, more detailed velocity field, turbulence, and void fraction data and analysis have been investigated in recent studies (Blenkinsopp and Chaplin, 2007; Deane and Stokes, 2002; Drazen and Melville, 2009; Grue and Jensen, 2006; Kimmoun and Branger, 2007; Melville et al., 2002). Due to the technical difficulties, the experimental measurements can only be done in the water region, and detailed description of the flow field in the energetic wave breaking region is not available. With the development of the computational fluid dynamics (CFD) technology, detailed wave breaking process and velocity profile can be obtained in both water and air phases (Chen et al., 1999; Watanabe and Saeki, 2002). The early CFD studies are

* Corresponding author. Tel.: +1 319 335 5215; fax: +1 319 335 5238.
E-mail address: frederick-stern@uiowa.edu (F. Stern).

0889-9746/$ - see front matter
doi:10.1016/j.jfluidstructs.2011.10.011

usually 2-D due to the prohibitive computational cost for the 3-D simulations. In the more recent CFD studies (Iafrati, 2010; Lubin et al., 2006; Watanabe et al., 2005), simulations are conducted with the air entrainment, 2-D and 3-D vortex structures, and energy dissipation discussed.

It should be noted that most previous studies on plunging wave breaking are for deep water or sloping beach for which wave plunges forward in the same direction of the mean flow. Yao and Wu (2005) experimentally investigated the shear currents effects on unsteady waves but with a focus on incipient breaking. Moreover, the geometry and conditions in most cases of CFD differ from the experiments even though the experiments are usually used to guide the analysis of CFD. Present interest is ship hydrodynamics for which body–wave interactions are important and the direction of wave breaking is opposite or at angle to the mean flow. Previous research used model ships in towing tanks focused on scars, vortices and mean and root mean square (rms) wave elevation induced by ship bow and shoulder wave breaking (Dong et al., 1997; Miyata and Inui, 1984; Olivieri et al., 2007), which suggests the presence of underlying coherent structures. A complementary CFD study to the latter study was carried out by Wilson et al. (2007). However, typical plunging wave breaking can hardly be obtained using model ships in towing tanks, and detailed measurements of the wave breaking processes are difficult. Recently, Shakeri et al. (2009) provide detailed measurements and analysis of divergent bow waves using a unique wave maker for simulating 2D+t flow. A numerical study using a 2D+t model has been reported by Marrone et al. (2011) for high speed slender ships. For slender bow ships, 2D+t wave breaking process is similar to deep water and sloping beach studies, i.e., plunges with forward splash-ups. In the early experimental studies by Ducan (1981, 1983), a fully submerged, two-dimensional hydrofoil was towed horizontally to produce breaking waves. These studies are focused on spilling breakers. Greco et al. (2004) investigated the impact flows on ship-deck structures due to head-incoming waves.

In the experimental fluid dynamics (EFD) study by Kang et al. (in this issue, hereafter referred to as Part I), a quadratic profile bump mounted in a shallow water flume is used to create impulsive sub critical flow conditions where plunging wave breakers are successfully obtained. Ensemble-averaged measurements (relative to the time t_b at which the maximum wave height is reached just before the first plunge) are conducted, including the overall flume flow and 2-D PIV center-plane velocities and turbulence inside the plunging breaking wave and bottom pressures under the breaking wave. The plunging wave breaking that is triggered by the flow over a submerged bump is of relevance to ship hydrodynamics since it includes wave–body interactions and the wave breaking direction is opposite to the mean flow as discussed in Part I. The idea and approach of creating plunging wave breakers using a submerged bump is obtained collectively from the previous experimental (Cahouet, 1984; Miyata et al., 1985) and CFD studies (Huang et al., 2007; Iafrati et al., 2001; Yang and Stern, 2007). The CFD results were used as a guide for the test design of the experiments (Ghosh, 2008) and the initial experimental data was used for validation. Subsequently, a complementary CFD study was used to aid in the data analysis simultaneously as the experimental data is used to validate a Cartesian grid, immersed boundary, coupled level set, and volume-of-fluid CFD method (Wang et al., 2009). Wang et al. (2009) indentified three repeated plunging events each with three sub-events [jet impact (plunge), oblique splash, and vertical jet]; however, they used fully impulsive initial conditions and adjusted the initial velocity and water elevation to match Ghosh's (2008) wave breaking position, which precluded detailed temporal validation.

In the present part of this paper, impulsive plunging wave breaking downstream of a bump in a shallow water flume is numerically simulated with the aim of providing a detailed quantitative description of the overall plunging wave breaking process. The time-dependent velocity and wave elevation boundary conditions are specified at the inlet and outlet using the exact experimental data provided in Part I. The computational results are compared with the experimental measurements to validate the capability of the code of CFDShip-Iowa Version 6 (Yang and Stern, 2009; Wang et al., 2009) for wave breaking. The simulations are carried out using a Cartesian grid solver with the sharp interface, coupled level set and volume-of-fluid (CLSVOF) and immersed boundary methods.

2. Computational methods

CFDShip-Iowa Version 6, a sharp interface Cartesian grid solver for two-phase incompressible flows recently developed at IIHR by Yang and Stern (2009), is used for the computational study. In this solver, the interface is represented by the level set (LS) method, which was later extended by Wang et al. (2009) using a coupled level set and volume-of-fluid (CLSVOF) method. A ghost fluid methodology is adopted to handle the jump conditions across the interface, where the density and surface tension effect are treated in a sharp way while the viscosity is smeared by a smoothed Heaviside function. A sharp interface embedded boundary method is used to handle complex immersed boundaries on Cartesian grids (Yang and Balaras, 2006).

2.1. Mathematical model

For the incompressible viscous flows of two immiscible fluids with constant properties, the Navier–Stokes equations are given by

$$\frac{\partial \mathbf{u}}{\partial t}+\mathbf{u}\cdot\nabla\mathbf{u}=\frac{1}{\rho}\nabla\cdot(-p\mathbf{I}+\mathbf{T})+\mathbf{g}, \quad (1)$$

$$\nabla\cdot\mathbf{u}=0, \quad (2)$$