Ocean Engineering 128 (2016) 105–115

Contents lists available at ScienceDirect

Ocean Engineering

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

Breaking wave interaction with a vertical cylinder and the effect of breaker location

Arun Kamath*, Mayilvahanan Alagan Chella, Hans Bihs, Øivind A. Arntsen

Department of Civil and Transport Engineering, Norwegian University of Science and Technology (NTNU), 7491 Trondheim, Norway

ARTICLE INFO

Keywords:
Breaking wave
Wave forces
Wave impact
Vertical cylinder
Computational Fluid Dynamics
REEF3D

ABSTRACT

The open-source CFD model REEF3D is used to simulate plunging breaking wave forces on a vertical cylinder. The numerical results are compared with data from the experiments carried out at the Large Wave Channel, Hannover, Germany to validate the model. Further, the location of the cylinder is changed so that the breaking wave impacts the cylinder at different stages of wave breaking and the resulting wave forces are evaluated. The different locations for the cylinder placement based on the breaker location are determined from the results obtained for the wave breaking process in a two-dimensional numerical wave tank. Maximum wave forces are found to occur when the breaking wave tongue impacts the cylinder just below the wave crest in all the cases simulated and the lowest wave forces are generally obtained when the wave breaks behind the cylinder. Several wave features such as the splashing on impact, the splitting and rejoining of the wave around the cylinder resulting in a chute-like jet formation are identified. The model provides a good representation of the breaking wave process and can be a useful tool to evaluate breaking wave forces on structures.

1. Introduction

A lot of research work has been carried out in the past on the evaluation of wave forces on structures exposed to waves due to their importance in coastal and offshore engineering. Wave interaction with a vertical circular cylinder depends on the Keulegan-Carpenter (KC) number and the relative size of the cylinder with respect to the incident waves. The KC number is a ratio between the excursion length of the fluid particles to the length of the obstacle in the flow. In the case of vertical circular cylinders in a wave field, it is given by $KC = uT/D$, where u is amplitude of the horizontal fluid velocity, T is the wave period and D is the diameter of the cylinder (Sumer and Fredsøe, 1997). The ratio measures the importance of the inertial forces and the drag forces. The wave forces on cylinders at higher KC numbers ($KC > 2$) and cylinder diameter to wavelength ratio $D/L < 0.2$ are generally determined using the Morison formula (Morison et al., 1950) to account for inertial and drag component of the wave forces using empirical force coefficients. In the case of breaking wave forces, the Morison formula cannot be directly applied because breaking waves are associated with impact forces of very high magnitudes acting over a short duration. In order to describe the total force from breaking waves with the Morison equation, an impact force term is considered in addition to the quasi-static forces (Goda et al., 1966). Present knowledge concerning the breaking wave forces is gained from experiments by Goda et al. (1966), Wienke and Oumeraci (2005), Arntsen et al. (2011) to name a few, but the measurement of velocity and acceleration under breaking waves and their interaction with structures is very demanding. The theoretical description of the impact force involves the use of several parameters such as slamming coefficients, curling factor, breaker shape and wave kinematics at breaking which have to be determined experimentally. Previous studies on breaking wave forces such as Chan and Melville (1988), Bullock et al. (2007), Wienke and Oumeraci (2005) have indicated that breaking wave impact characteristics depend on several parameters such as the depth inducing breaking, breaker type and the distance of the structure from the breaker location.

The modeling of breaking waves in shallow waters is challenging due to the complex nature of the physical processes including highly non-linear interactions. A considerable amount of numerical studies have been attempted to model wave breaking over plane slopes (Lin and Liu, 1998; Zhao et al., 2004; Alagan Chella et al., 2015b). These studies have helped extend the knowledge regarding breaking wave characteristics and the geometric properties of breaking waves. The quantification of these breaking wave parameters are an important input to improve the empirical coefficients used for the evaluation of breaking wave forces. Though many extensive numerical studies exist in current literature that study the wave breaking process, not many have been extended to study the forces due to breaking waves and the

---

* Corresponding Author.
E-mail address: arun.kamath@ntnu.no (A. Kamath).

http://dx.doi.org/10.1016/j.oceaneng.2016.10.025
Received 12 May 2015; Received in revised form 26 September 2016; Accepted 12 October 2016

effect of breaker types on the wave forces. Bredmose and Jacobsen (2010) studied breaking wave impact forces due to focussed waves with the Jonswap wave spectrum for input and carried out computations for half the domain assuming lateral symmetry of the problem using OpenFOAM. Mo et al. (2013) measured and modelled solitary wave breaking and its interaction with a slender cylinder over a plane slope for a single case using the filtered Navier-Stokes equations with large eddy simulation (LES) turbulence modeling, also assuming lateral symmetry and showed that their numerical model sufficiently captured the important flow features. Choi et al. (2015) investigated breaking wave impact forces on a vertical cylinder and two cases of inclined cylinders for one incident wave using the modified Navier-Stokes equations with the volume of fluid (VOF) method for interface capturing to study the dynamic amplification factor due to structural response.

The study of breaking wave forces using computational fluid dynamics (CFD) can provide a very detailed description of the physical processes as the fluid physics are calculated with few assumptions. With high-order discretization schemes for the convection and time advancement, sharp representation of the free surface and tight velocity-pressure coupling in the model, the wave transformation, wave hydrodynamics and flow features can be represented very accurately and in a realistic manner. In the complex case of breaking wave interaction with structures, CFD simulations can be used to capture the details of the flow field that are challenging to capture in experimental studies due to various factors including cost, instrumentation and structural response. Different wave loading scenarios can be analysed as the breaker locations are easier to analyse and maintain in the simulations.

In the current study, the open source CFD model REEF3D (Bihs et al., 2016) is used to simulate periodic breaking wave forces on a slender cylinder in a three-dimensional wave tank without assuming lateral symmetry. The model has been previously used to simulate the wave breaking process under different conditions (Alagan Chella et al., 2015a, 2015c) and the wave breaking kinematics were fully represented including the motion of the jet, air pocket formation and the reconnection of the jet with the preceding wave trough. The model provides a detailed representation of the free surface and is numerically stable for various problems related to wave hydrodynamics. It is fully parallelised, has shown very good scaling on the high performance computing system at NTNU provided by NOTUR (2012) and can be used to carry out complex simulations efficiently on a large number of processors.

This paper presents the breaking wave interaction with a vertical cylinder. Three different wave heights are simulated and the evolution of wave breaking over a 1:10 slope is studied using two-dimensional simulations. The locations for the placement of the cylinder to investigate five different wave loading cases based on Irschik et al. (2002) are identified from these two-dimensional studies. Next, the wave forces in the different scenarios for the three different incident wave heights are evaluated in a three-dimensional numerical wave tank. The numerical model is validated by comparing the calculated wave forces and the free surface with experimental data from experiments carried out in the Large Wave Channel (GWK), Hannover, Germany. The wave interaction with the vertical cylinder in selected two different scenarios is investigated and the effect of the cylinder placement with respect to the breaker location on the free surface features is presented.

2. Numerical model

The open-source CFD model REEF3D solves the fluid flow problem using the incompressible Reynolds-Averaged Navier-Stokes (RANS) equations along with the continuity equation:

$$\frac{\partial u_i}{\partial x_i} = 0 \tag{1}$$

$$\frac{\partial u_i}{\partial t} + u_j \frac{\partial u_i}{\partial x_j} = -\frac{1}{\rho}\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j}\left[(\nu + \nu_t)\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right)\right] + g_i \tag{2}$$

where u is the velocity averaged over time t, ρ is the fluid density, p is the pressure, ν is the kinematic viscosity, ν_t is the eddy viscosity and g is the acceleration due to gravity.

The pressure is determined using Chorin's projection method (Chorin, 1968) and the resulting Poisson pressure equation is solved with a preconditioned BiCGStab solver (van der Vorst, 1992). Turbulence modeling is handled using the two-equation $k - \omega$ model proposed by Wilcox (1994), where the transport equations for the turbulent kinetic energy, k and the specific turbulent dissipation rate, ω are:

$$\frac{\partial k}{\partial t} + U_j \frac{\partial k}{\partial x_j} = \frac{\partial}{\partial x_j}\left[\left(\nu + \frac{\nu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right] + P_k - \beta_k k\omega \tag{3}$$

$$\frac{\partial \omega}{\partial t} + U_j \frac{\partial \omega}{\partial x_j} = \frac{\partial}{\partial x_j}\left[\left(\nu + \frac{\nu_t}{\sigma_\omega}\right)\frac{\partial \omega}{\partial x_j}\right] + \frac{\omega}{k}\alpha P_k - \beta\omega^2 \tag{4}$$

$$\nu_t = \frac{k}{\omega} \tag{5}$$

where, P_k is the production rate and closure coefficients $\sigma_k = 2$, $\sigma_\omega = 2$, $\alpha = 5/9$, $\beta_k = 9/100$, $\beta = 3/40$.

The highly strained flow due to the propagation of waves in the tank results in an overproduction of turbulence in the numerical wave tank as the eddy viscosity is determined from the strain in the convective terms. The Bradshaw et al. (1967) assumption is used to limit the eddy viscosity as shown by Durbin (2009):

$$\nu_t \le \sqrt{\frac{2}{3}}\frac{k}{|\mathbf{S}|} \tag{6}$$

where $\mathbf{S}$ stands for the source terms in the transport equations. In a two-phase CFD model, the large difference between the density of air and water leads to a large strain at the interface, which leads to an overproduction of turbulence at the free surface. In reality, the free surface is a boundary at which eddy viscosity is damped naturally which the standard $k - \omega$ model does not account for. In order to avoid the overproduction of turbulence at the free surface, the specific turbulence dissipation at the free surface is defined using the empirical relationship presented by Naot and Rodi (1982).

The discretization of the convective terms of the RANS equations are discretized using the fifth-order conservative finite difference Weighted Essentially Non-Oscillatory (WENO) scheme (Jiang and Shu, 1996). The Hamilton-Jacobi formulation of the WENO scheme (Jiang and Peng, 2000) is used to discretize the level set function ϕ, turbulent kinetic energy k and the specific turbulent dissipation rate ω. The WENO scheme is at minimum a third-order accurate scheme in the presence of large gradients and provides sufficient accuracy required to model complex free surface flows. The time advancement of the momentum equation, the level set function and the reinitialisation equation is treated with a Total Variation Diminishing (TVD) third-order Runge-Kutta explicit time scheme (Shu and Osher, 1988). The Courant-Frederick-Lewis (CFL) criterion is maintained at a constant value throughout the simulation using an adaptive time stepping strategy to determine the time steps. A first-order implicit scheme for the time advancement of k and ω removes the large source term contributions from these variables for the evaluation of the CFL criterion. This is reasonable, as these variables are largely driven by source terms and have a low influence from the convective terms. The diffusion terms of the velocities are also handled using an implicit scheme, removing them from the CFL criterion and the maximum velocities in the domain are used to determine the time steps to