Journal of Fluids and Structures 75 (2017) 99–116

Contents lists available at ScienceDirect

Journal of Fluids and Structures

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

Investigations of offshore breaking wave impacts on a large offshore structure

Zheng Zheng Hu *, Tri Mai, Deborah Greaves, Alison Raby

School of Marine Science and Engineering Faculty of Science and Environment, University of Plymouth, Plymouth, Devon, PL4 8AA, UK

HIGHLIGHTS

- Wave impact types for a truncated wall are similar to those for a full depth wall.
- A second-order NewWave inlet boundary condition is implemented in the NWT.
- Four distinct wave impact types with the same input energy are generated.
- Numerical simulations of run up and peak pressure agree well with experiments.
- The highest force on the hull occurs under the large air pocket wave impact.
- The highest run-up at the hull occurs for the slightly-breaking wave impact.
- The highest pressure recorded at any location was due to flip-through impact.
- Different wave impact types should be considered for different design purposes.

ARTICLE INFO

Article history:
Received 22 March 2016
Received in revised form 8 August 2017
Accepted 9 August 2017

Keywords:
OpenFOAM
Focused wave groups
Breaking waves
Nonlinear $k-\epsilon$ model
Air entrainment
Impact pressure

ABSTRACT

This paper describes numerical and laboratory investigations that have been carried out to gain a better understanding of the physical processes involved in offshore breaking wave impacts on a large offshore structure. The findings are relevant to offshore and coastal structures and to identifying the extreme loads, peak pressures and maximum run-up needed for their design. A truncated wall in a wave flume is used to represent a vertical section of an FPSO (Floating Production Storage and Offloading) hull, which is a typical large offshore structure. Four types of wave impact were identified in the tests, and are referred to as slightly-breaking, flip-through, large air pocket and broken wave impacts. Physical modelling was undertaken in Plymouth University's COAST Laboratory and the open source Computational Fluid Dynamics (CFD) package-Open Field Operation and Manipulation (OpenFOAM) was adopted to study focused wave generation and wave impact on the hull. The method solves incompressible Unsteady Reynolds-averaged Navier–Stokes Equations (URANSE) using a finite volume method with two phase flows. A Volume of Fluid (VoF) interface capturing approach is used to model the free surface. A NewWave boundary condition is used to generate focused wave groups based on the first plus second-order (hereafter second-order) Stokes wave theory in the Numerical Wave Tank (NWT). By changing the focus location with respect to the wall, the wave impact type was altered in both the numerical and laboratory investigations.

The results show that for the four wave impact types tested good agreement was achieved between numerical predictions and experimental measurements of surface elevation, run up and impact force. The peak pressures predicted by the simulation are lower than the experimentally measured results due to time step constraints, although the shape of the pressure time history is very similar. Four distinct wave impact types are identified

* Corresponding author.
E-mail address: zheng.hu@plymouth.ac.uk (Z.Z. Hu).

http://dx.doi.org/10.1016/j.jfluidstructs.2017.08.005

for the vertical hull section and are found to be similar in character to those observed for a full depth vertical wall. The predicted force on the hull is found to be greatest for the large air pocket impact, and the highest run-up for the slightly-breaking wave impact. The pressure records show a high degree of spatial and temporal variation though the highest pressure recorded at any location was due to flip-through. This research has shown that different characteristic wave impact types are responsible for maximum load and greatest wave run-up and so need to be considered separately for design purposes.

1. Introduction

Large offshore structures, ships, storm surge barriers and closure dams can have vertical faces that are exposed to wave impact. Such impacts can lead to very high peak pressures and wave loads, which may lead to structural failure, and are likely to be more common as storm occurrence increases due to climate change. The importance of investigating wave loading on offshore structures and in particular the loads and run-up due to wave impact under a changing climate is being recognised to an increasing extent. For example, Tanimoto and Takahashi (1994) described how an extreme wave impact can displace large caissons by several metres, and design formulae for estimating the magnitude of the impulsive pressure generated by breaking waves have recently been presented by Goda (2000) and Oumeraci et al. (2001). Guilcher et al. (2013, 2014) used the SPH (Smoothed Particle Hydrodynamics) method to model a unidirectional breaking wave impacting a rigid wall and compared the numerical predictions with experimental results under two different scales (full scale and scale 1:6). In this paper, a truncated vertical wall is used to represent a vertical section of the hull of a fixed FPSO and investigated to provide understanding of the underlying processes characterised by four wave impact types.

For coastal structures, accurate prediction of the most severe wave loading is crucial to their design. In laboratory studies, Bullock et al. (2007) investigated the detailed characteristics of four different types (slightly-breaking, low-aeration, high-aeration and broken) of wave impact on a vertical and sloping wall, and found that the impacts depend on the breaker conditions. Bredmose et al. (2010) found that the flip-through impact can generate very large localised pressure on the wall without any hint of entrained or trapped air, and Bredmose et al. (2015) studied the effects of scale and aeration on two impact types (flip-through and low-aeration impacts), by physical and numerical investigation. Wave impacts are very sensitive to the wave shape just before impact. For example, an air pocket may be trapped if the wave overturns as it strikes the wall, and large quantities of air can be entrained during breaking so that a turbulent air–water mixture strikes the wall when the wave has already broken. In either process, the compressibility of the trapped or entrained air will affect the dynamics, reduce the maximum pressure due to a cushioning effect, but tend to distribute the impact pressure more widely so that the overall force on the wall may not necessarily be reduced (see Peregrine et al., 2005). The presence of air may extend the duration of peak pressure and lead to rebound that can increase the total impulse on the structure (see Wood et al., 2000).

Dold (1992) and Cooker and Peregrine (1990) investigated the wave impact problem numerically by means of fully nonlinear potential flow theory, which assumes incompressible, inviscid single phase fluid flow and has been used to model wave propagation over an elliptical mound and onto a vertical wall. This gives valuable insight into the way high pressures are generated but the model suffers from the limitation of only being applicable to non-aerated flows up to a point just after the waves overturn. Bredmose et al. (2010) used nonlinear potential flow to study the flip-through wave impact on a wall but the numerical computation broke down during the impact while the vertical jet was emerging. Colicchio et al. (2007) applied a level-set method and an SPH method to model an impact that traps a small pocket of air. Khayyer and Gotoh (2009) developed several modified forms of the MPS (Moving Particle Semi-implicit) method which they compared with the results of Hattori et al. (1994) for a flip-through wave impact. The AMAZON-SC 3D code (see Hu et al., 2013a, b) used a high resolution Godunov-type approach for spatial discretisation of the Euler and Navier–Stokes equations and employed both air and water fluid regions for 3D water impact problems. It uses an exact Riemann solver for hyperbolic conservation laws, which enables shock waves to be reproduced. Two software packages CFX and STAR CCM+ were validated against measured results for an oscillating cone at the water surface by Westphalen et al. (2009), in which the fully nonlinear Navier–Stokes equations were solved by a control volume Finite Element (CV-FE) and a Finite Volume (FV) method in laminar fluid flow. In this paper, the incompressible Unsteady Reynolds-averaged Navier–Stokes Equations (URANSE and hereafter RANS) in OpenFOAM (hereafter OF) are solved using a finite volume method in two phase flows and the Nonlinear k–ϵ model developed by Brown et al. (2014, submitted for publication) is selected for this study because the turbulence model becomes important in cases of wave breaking as considered here. A Volume of Fluid (VoF) interface capturing approach is used to model the free surface, which can handle break-up, overturning and recombination, including the flow of the surrounding air and entrainment of air pockets at the hull. In this way, consideration of effects including viscosity, aeration and turbulence may be taken into account during simulation of violent wave impact. The only drawback of the study is that hydroelasticity and the compressibility of water and air is not considered in the flow solution and the effect of this omission on the quality of the results is discussed.

The impacting waves were generated using a focused wave group technique, which enables a highly nonlinear extreme wave to be produced on demand at both a given time and location. Focused wave groups have been used for impact