Coastal Engineering 125 (2017) 81–94

Contents lists available at ScienceDirect

Coastal Engineering

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

Application of a buoyancy-modified *k-ω SST* turbulence model to simulate wave run-up around a monopile subjected to regular waves using OpenFOAM®

Brecht Devolder[a,b,*], Pieter Rauwoens[b], Peter Troch[a]

[a] Ghent University, Department of Civil Engineering, Technologiepark 904, 9052 Ghent, Belgium
[b] KU Leuven, Department of Civil Engineering, Technology Cluster Construction, Zeedijk 101, 8400 Ostend, Belgium

ARTICLE INFO

Keywords:
CFD
OpenFOAM®/IHFOAM
Buoyancy-modified turbulence model
Monopile
Wave run-up

ABSTRACT

The objective of the present work is to investigate wave run-up around a monopile subjected to regular waves inside a numerical wave flume using the Computational Fluid Dynamics (CFD) toolbox OpenFOAM®. Reynolds-Averaged Navier-Stokes (RANS) turbulence modelling is performed by applying the *k-ω SST* model. Boundary conditions for wave generation and absorption are adopted from the IHFOAM toolbox. Simulations of propagating water waves show sometimes excessive wave damping (i.e. a significant decrease in wave height over the length of the numerical wave flume) based on RANS turbulence modelling. This anomaly is prevented by implementing a buoyancy term in the turbulent kinetic energy equation. The additional term suppresses the turbulence level at the interface between water and air. The proposed buoyancy-modified *k-ω SST* turbulence model results in an overall stable wave propagation model without significant wave damping over the length of the flume. Firstly, the necessity of a buoyancy-modified *k-ω SST* turbulence model is demonstrated for the case of propagating water waves in an empty wave flume. Secondly, numerical results of wave run-up around a monopile under regular waves using the buoyancy-modified *k-ω SST* turbulence model are validated by using experimental data measured in a wave flume by De Vos et al. (2007). Furthermore, time-dependent high spatial resolutions of the numerically obtained wave run-up around the monopile are presented. These results are in line with the experimental data and available analytical formulations.

1. Introduction

Numerous offshore wind farms have already been installed or are under construction. Wind turbines are mounted on large foundations in the seabed, such as vertical cylinders, called monopiles. The design of such a monopile is mainly dependent on the total force acting on it. However, some smaller pieces are attached to that monopile (e.g. boat landing facility, J-tube, ladder, platform and door). In order to design these smaller parts, wave run-up around the monopile caused by incident waves should be assessed accurately. Therefore, both experimental and numerical research have already been conducted.

Experimental research has been performed in order to define the wave run-up pattern around a monopile. For example, De Vos et al. [5] described small scale model tests in which wave run-up was measured around a monopile placed in relatively deep water conditions using different regular and irregular wave trains. Moreover, analytical formulations are proposed to determine the maximum wave run-up for both regular and irregular waves. Kazeminezhad and Etemad-Shahidi [14] have recently re-analysed several datasets and presented alternative formulations in which pre-calculation of the wave kinematics is not necessary to assess the maximum wave run-up.

Numerical modelling of wave run-up around a monopile is also reported in literature. Christensen et al. [3] described a study of the forces acting on a monopile caused by extreme waves propagating over a sloping bed. Numerical results were compared with analytical solutions and experiments and a good agreement was found. A numerical study with a 3-D ComFLOW model performed by Peng et al. [22] reproduced experimental data measured by De Vos et al. [5]. A grid sensitivity study showed that a minimum grid size of $D/10$ was needed in the zones of interest (i.e. around the still water level and near the monopile with diameter D) to obtain a grid independent solution. The Courant-Friedrichs-Lewy (CFL) condition controlled the time step which may not exceed $T/100$ for accuracy purposes (where T is the wave period). The paper presented only absolute values of wave run-up and no comparisons with regard to time series were provided. The authors also mentioned that for large wave run-up, the numerical

* Corresponding author at: Ghent University, Department of Civil Engineering, Technologiepark 904, 9052 Ghent, Belgium.
E-mail addresses: Brecht.Devolder@UGent.be (B. Devolder), pieter.rauwoens@kuleuven.be (P. Rauwoens), Peter.Troch@UGent.be (P. Troch).

http://dx.doi.org/10.1016/j.coastaleng.2017.04.004
Received 24 June 2016; Received in revised form 4 January 2017; Accepted 10 April 2017

model slightly underestimates the measured run-up. Lara et al. [15] presented a numerical simulation of a pile group subjected to waves using the IHFOAM toolbox. Only numerical results were presented which indicated that IHFOAM is a capable toolbox for analysing wave run-up around and wave-induced forces on offshore piles. Ransley et al. [24] compared numerical results with experimental data for extreme wave impacts on a fixed truncated circular cylinder. The numerical solution was obtained without turbulence modelling but the authors expect that it plays an important role in (extreme) wave-structure interaction. El Safti et al. [6] presented a hybrid 2D-3D CFD model to investigate wave forces on piled structures. In this study, turbulent effects were incorporated by using a one-equation eddy-viscosity Sub-Grid Scale (SGS) Large Eddy Simulation (LES) model. The authors' motivation to use LES was to include the effects of air compressibility during breaking wave impact on structures. Paulsen et al. [21] analysed strong nonlinear forces caused by steep or breaking waves and ringing loads due to steep nonlinear waves. Turbulence modelling was excluded because the forces acting on the monopile were mainly inertia dominated. A fair agreement was found between numerical and experimental data. Chen et al. [2] investigated nonlinear wave interactions with offshore structures for different wave conditions. They concluded that OpenFOAM® is suitable for accurate modelling of nonlinear wave interactions with monopiles. The time step was initially 0.01 s and changed automatically to satisfy a maximum Courant number of 0.5. Grid convergence was reached for a horizontal and vertical resolution of respectively $L/70$ and $H/8$ (with a refinement factor of 2 around the still water level and the monopile; where L is the wave length and H is the wave height).

More recently, Kamath et al. [12,13] reported CFD results of wave interaction with multiple vertical cylinders. They performed simulations using a k-ω turbulence model and observed unphysical wave damping based on RANS turbulence modelling. Therefore, both eddy viscosity limiters and free surface turbulence damping at the interface were applied. This unphysical wave damping caused by RANS turbulence modelling is not only observed during CFD simulations of monopiles. Several other authors also reported wave damping when using CFD for wave modelling: Mayer and Madsen [18], Jacobsen et al. [11], Vanneste and Troch [30] and Elhanafi et al. [7].

In general, the majority of literature presents wave-induced forces acting on a monopile rather than wave run-up phenomena. Therefore, turbulence modelling was omitted and no indication of the influence of turbulence on wave run-up was given. However, some authors reported the necessity of using a turbulence model. For example Higuera et al. [8,9] applied both k-ε and k-ω *SST* turbulence models since they are widely used. Furthermore, turbulence modelling is needed in the case of significant vortex shedding or when wave breaking occurs around the monopile due to even steeper waves. This can happen when irregular waves are generated, then energy is transferred between the different frequencies increasing the wave height at a particular time instant and at a certain location. If that location is close to the monopile, waves can break and prominent vortex shedding can occur. This paper will tackle the implementation of a suitable turbulence model in order to simulate properly wave propagation in a numerical wave flume and wave run-up around a monopile. In general, non-breaking waves induce a very low level of turbulence. However, when the wave-induced flow encounters an object, a monopile in this case, the flow is disturbed and becomes turbulent. For the waves studied in this research, the Reynolds (Re) number and Keulegan-Carpenter (KC) number vary from $4.65 \cdot 10^4$ to $5.84 \cdot 10^4$ and from 4.26 to 5.17 respectively. According to Sumer and Fredsøe [27], a pair of asymmetric vortices will develop resulting in a turbulent flow around the monopile. Moreover, even if the KC numbers are small for the waves studied, the boundary layer around the monopile may be turbulent.

We present a study of wave run-up around a monopile subjected to regular waves using the Computational Fluid Dynamics (CFD) toolbox OpenFOAM® [20]. Reynolds-Averaged Navier-Stokes (RANS) turbulence modelling is performed by applying the k-ω *SST* model. Sometimes, this RANS approach causes excessive wave damping (i.e. a significant decrease in wave height over the length of the numerical wave flume). Therefore, a buoyancy term is implemented in the turbulent kinetic energy (TKE) equation of the k-ω *SST* model. The idea of adding a buoyancy term is taken from Van Maele and Merci [29] who modify the k-ε model to simulate buoyant plumes. These fire-induced flows are characterised by continuous density variations. For water waves, the density is discontinuous at the free water surface resulting in an infinite density gradient. However, when a Volume of Fluid (VoF) method is applied for wave modelling, the density gradient is smeared out over several cells leading to a continuous change in density around the air-water interface. Consequently, the change in density around the interface between water and air is similar to the change in density observed in fire flows. As a result of implementing a buoyancy term, an overall stable wave propagation model without significant wave damping over the length of the flume is obtained. Numerical simulations are performed and compared with two different sets of wave parameters described in the laboratory study of De Vos et al. [5].

The remainder of this paper is organised as follows. Firstly, in Section 2, the previous experimental study by De Vos et al. [5] is introduced. In Section 3, the governing equations for the numerical model are presented, followed by a description of the computational domain, the boundary conditions applied and the solver settings. Subsequently in Sections 4 and 5, the numerical model is used to perform several simulations while in Section 6 the obtained results are discussed in detail. Finally, the conclusions and future work are drawn in Section 7.

2. Previous experimental study

In this research, data is reused from a laboratory study by De Vos et al. [5] conducted in a wave flume at Aalborg University, Denmark. The flume has a length of 30 m, a width of 1.5 m and a height of 1 m. The pile diameter was 0.12 m whereas a constant water depth of 0.50 m was maintained during regular wave tests. The offshore slope was held constant at 1:100 and will be neglected in the present numerical study because shoaling effects are negligible.

A definition sketch of the wave gauges' position to measure wave run-up around the monopile is given in Fig. 1. Herein, nine wave gauges are installed 2 mm away from the monopile's surface. The position of the wave gauges is an important parameter to determine wave run-up. Therefore, the position of the numerical wave gauges is the same as the ones installed in the experimental flume. The nine wave gauges are characterised by their angle with respect to the incoming waves of respectively 0°, 22.5°, 45°, 67.5°, 90°, 112.5°, 135°, 157.5° and 180°, as depicted in Fig. 1.

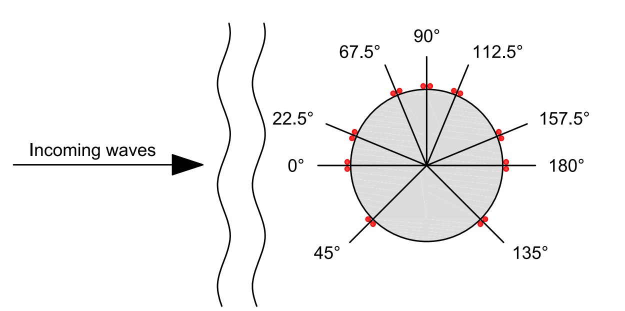

Fig. 1. Position of the nine wave gauges (two dots per gauge) around the monopile where each wave gauge is characterised by its angle with respect to the direction of the incoming waves. The wave gauges are installed 2 mm away from the monopile's surface (adapted from [5]).