Ocean Engineering 137 (2017) 78–88

Contents lists available at ScienceDirect

Ocean Engineering

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

A comparison of numerical simulations of breaking wave forces on a monopile structure using two different numerical models based on finite difference and finite volume methods

Jithin Jose[a,*], Sung-Jin Choi[b], Knut Erik Teigen Giljarhus[c], Ove Tobias Gudmestad[a]

[a] Department of Mechanical and Structural Engineering and Materials Science, University of Stavanger, Norway
[b] Loads Copenhagen, DNV GL, Copenhagen, Denmark
[c] Lloyd's Register, Stavanger, Norway

ARTICLE INFO

Keywords:
Finite volume method
Finite difference method
Wave breaking
Secondary loads
Monopile
Volume of fluid method

ABSTRACT

The nonlinear forces from breaking waves are a major concern in the design of offshore structures. Due to the complexity of the wave-breaking phenomenon, understanding the interaction of breaking waves with a structure is always a challenging task. The use of numerical models can be a useful tool for studying such a phenomenon. At present, many numerical models are available, using either a Finite Difference Method (FDM) or a Finite Volume Method (FVM), for solving the governing equations. In wave breaking studies, different researchers have come up with reasonable results, using both models. However, there have been few attempts to compare the relative strengths and weaknesses of the two methods. In the present paper a comparison of both methods applied to breaking wave studies is performed. Two different 3D Navier-Stokes solvers, 2PM3D (FDM) and OpenFOAM (FVM), are used to simulate the breaking wave forces on a monopile structure. Two different scenarios are considered for generating non-breaking and breaking waves, and the results are compared with theoretical results and available experimental measurements. For both numerical models, the breaking wave interactions with the monopile were in good agreement with the experimental measurements.

1. Introduction

In real sea, offshore structures are subjected to nonlinear wave interactions, such as wave breaking and green water impact. These nonlinear wave interactions sometimes result in damage to offshore structures. Therefore, understanding these phenomena is very important for the design of offshore structures. The forces from breaking waves have been a major concern for offshore structures installed in shallow waters, and these breaking wave impact forces sometimes govern the overall design of such structures. The physical realization of breaking wave interactions with the structure is a challenging task, due to the complexity of the wave-breaking phenomenon and the time-dependent shape of the breaking wave (Hull and Müller, 2002). Most previous studies on breaking waves focused on experimental measurements (Goda et al., 1966; Wienke and Oumeraci, 2005), and were limited to simple structures and specific experimental conditions.

The solution to these challenges could be the use of a well-validated numerical model, which can simulate breaking waves. The increase in computational capabilities and advanced numerical codes makes numerical modelling a powerful tool to predict the wave-breaking forces on structures. Moreover, these numerical models can estimate the wave forces by means of direct pressure integration over the structure, without using any empirical relations. However, the accuracy and efficiency of the numerical model depends on the numerical methods used.

Numerical models based on solving Navier Stokes equations are widely used to simulate breaking waves. There are two main classical methods for obtaining the solution to these differential equations, namely, the Finite Difference Method (FDM) and the Finite Volume Method (FVM). The former is based on the application of the Taylor series expansion to approximate the governing differential equations (Sherwin and Peiro, 2005). It uses a rectangular grid of lines to represent the discretization of the differential equations. In the Finite Volume Method, the integral form of the differential equation is considered. The governing quantities are conserved over a finite volume. Within both the FDM and the FVM, there are many options for the discretization of the various terms in the governing equation, which also contributes to the accuracy of both methods.

Hur and Mizutani (2003) used a Navier-Stokes solver, based on the Finite Difference Method, to study the wave forces on an asymmetric

* Corresponding author.
E-mail address: jithin.jose@uis.no (J. Jose).

http://dx.doi.org/10.1016/j.oceaneng.2017.03.045
Received 7 November 2016; Received in revised form 22 March 2017; Accepted 24 March 2017
0029-8018/ © 2017 Elsevier Ltd. All rights reserved.

structure installed over a submerged breakwater. The model combines the Volume of Fluid (VOF) method (Hirt and Nichols, 1981) and the porous body model to simulate nonlinear wave deformation. The Large Eddy Simulation (LES) model was used to calculate the turbulence in the flow. Lee et al. (2011) studied the wave interactions around two vertical cylinders, using a 3D Navier-Stokes solver based on the FDM. The VOF method was used to account for free surface tracking. The computed results showed a good agreement with the experimental measurements. Hu and Kashiwagi (2004) developed a FDM with a Constrained Interpolation Profile (CIP) algorithm to study violent wave interactions with the structure. The free surface is distinguished by a density function, which is solved using the CIP method. Park et al. (2003) developed a finite difference viscous Navier-Stokes solver to study the nonlinear wave forces and run-up, around a conical gravity-based structure. The nonlinear free surface was treated by the marker-density function technique. The numerical model gave a reliable estimate of the maximum wave loading and run-up including a series of higher harmonic components. Choi (2014) used a 3D numerical model based on the FDM to simulate the breaking wave interactions on a vertical cylinder pile. To resolve the air-water interface, the VOF method was utilized.

On the other hand, Christensen et al. (2005) used a numerical model, based on solving 3D Navier- Stokes equations using the Finite Volume Method, to study the extreme wave forces and wave run-up on a cylindrical pile during wave breaking. The free surface is resolved with a VOF technique. Mo et al. (2007) developed a 3D numerical model, based on the Navier-Stokes equations, to study the wave interactions on a vertical pile. The model used the FVM to solve the governing equations, and the results were compared with the experimental measurements from the Large Wave Flume in Hannover. The VOF method was employed to track the nonlinear free surface. Chella et al. (2016) simulated wave breaking over a sloping seabed, using a numerical model, based on Reynolds averaged Navier-Stokes equations coupled with the level set method. The simulation results showed reasonable agreement with the experimental measurements by Ting and Kirby (1996). Jacobsen et al. (2012) developed a wave generation toolbox, waves2Foam, integrated with the open-source library, OpenFOAM, which is based on the FVM. The applicability of the toolbox to generate and absorb waves was demonstrated by comparing the results with benchmark test cases. Paulsen (2013) coupled the OpenFOAM with a potential flow solver to study the wave interactions with a vertical cylinder. The coupled FVM model results showed good agreement with the experimental measurements.

Apart from the above-mentioned Eulerian approaches, there is another well-known method, based on the Lagrangian scheme, in which the fluid is treated as particles and the path of each individual particle is tracked. Gotoh and Sakai (1999) used a moving particle semi-implicit (MPS) method to simulate breaking waves in the numerical wave tank. The method avoids the use of any free surface tracking method, such as VOF, to obtain the fluid surface. Shao (2006) used a numerical model, based on the smoothed particle hydrodynamic method coupled with a k-ε turbulence model, to simulate spilling and plunging waves. The mesh-free numerical results showed very good agreement with the experimental measurements. In comparison with the Eulerian approach, the Lagrangian approach required more computational time (Gotoh and Sakai, 1999), as each particle in the flow is tracked and the particle number needs to be very large to ensure the stability of the flow. However, this approach is not within the scope of the present paper.

There are many available numerical models, which use either FDM or FVM for solving the Navier-Stokes equations. In the wave breaking studies, different researchers have obtained reasonable results, using either model. According to the authors' knowledge, however, there has been no attempt to compare the relative strengths and weaknesses of the two models. There is an ongoing debate on the adequacy of both models in the study of highly nonlinear physical phenomena like wave breaking. The present paper performs a comparison of the two methods, using two different 3D Navier-Stokes solvers, the 2PM3D solver (Lee, 2006), which is based on the FDM, and OpenFOAM with waves2Foam toolbox, which is based on the FVM. The 2PM3D model uses a rectangular grid system to discretize the governing equations in the computational domain. In this model, in order to account for the geometry in the fluid domain, the cut cell method is used. However, in OpenFOAM, an unstructured mesh with body fitted grid method is used to model the geometries in the fluid domain. In the present study, two different scenarios are considered for simulating breaking and non-breaking waves in the numerical wave tank. The numerical models are used to simulate the breaking wave forces on a monopile structure. The simulation results are compared with available theoretical results and experimental measurements from the hydraulic model tests previously undertaken by Irschik et al. (2004). Secondary loads are observed on the cylinder when the wave flows across the structure. The capability of both numerical models to simulate the secondary load cycle on the cylinder structure is also analysed.

2. Model description

2.1. Experimental set-up

The experiment was carried out at the Large Wave Flume in Hannover, Germany (Irschik et al., 2004; Choi et al., 2015). The wave flume is 300 m long, 5 m wide and 7 m high. The slope at the bottom of the channel is 1/10. A cylindrical structure of 0.7 m diameter and 5 m length was erected at the edge of the slope. The pile was supported at the top and bottom by a transverse frame. During the experiments, there were two strain gauges integrated to the top and bottom of the monopile structure to measure the forces acting on the structure. The total breaking wave forces were calculated as the sum of the forces measured by these top and bottom transducers. The monopile structure was tested for a number of incident wave conditions for different orientations (vertical and inclined) of the pile. The waves were generated by a piston-type wave generator. Wave gauges and Acoustic Doppler Velocimeters (ADV) were distributed along the channel in order to track the wave surface elevation and water particle velocities, respectively. More details of the experimental set-up are given in Irschik et al. (2004).

2.2. Numerical model

In the present study, two different numerical models were used to simulate breaking waves in the computational domain. The numerical model 2PM3D, based on the FDM, was previously validated by Lee et al. (2011) and Choi et al. (2015). The OpenFOAM model with waves2Foam toolbox uses the FVM for solving the governing equations. Although the governing equations for both numerical models are the same, the method for solving the equations and treating the geometries in the computational domain differ. The numerical description of each model is provided in the following sections.

2.2.1. 2PM3D model

In order to study the breaking wave interactions with the structure, a numerical wave tank method developed by Lee (2006) was used. It comprises an internal wave source, an artificial damping zone to prevent the wave reflections at the lateral boundaries and a surface tracking function to treat the free surface. Assuming the two fluids are viscous, immiscible and incompressible, the fluid flow is governed by the continuity equation and the modified Navier-Stokes equation.

$$\frac{\partial(mv_j)}{\partial x_j} = q^* \tag{1}$$

Download English Version:

https://daneshyari.com/en/article/5474237

Download Persian Version:

https://daneshyari.com/article/5474237

Daneshyari.com