Coastal Engineering 102 (2015) 13–29

Contents lists available at ScienceDirect

Coastal Engineering

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

Numerical analysis of the interaction of irregular waves with two dimensional permeable coastal structures

Niels G. Jacobsen *, Marcel R.A. van Gent, Guido Wolters

Coastal Structures and Waves, Deltares, Delft, The Netherlands

ARTICLE INFO

Article history:
Received 5 January 2015
Received in revised form 16 April 2015
Accepted 11 May 2015
Available online 27 May 2015

Keywords:
Permeable structures
Irregular waves
Validation
Internal setup
Impact forces
OpenFoam/waves2Foam

ABSTRACT

This paper will address the validation and application of a volume of fluid method for coastal structures under the influence of normal incident irregular wave fields. Several physical processes will be addressed as part of the validation process, namely: (i) wave reflection from permeable and impermeable structures, (ii) wave transformation over a small shoal, (iii) wave damping inside of a permeable structure, (iv) the resulting wave induced internal setup and (v) wave induced forces. The numerical model will be validated against a multiple of experimental data sets for two dimensional coastal problems. The impact of air-relief gaps on the modelled wave induced pressures on a crest wall is analysed for the two dimensional layout of the structure. This is analysed to study the importance of the cushion effect from the incompressible air phase. Subsequent to the validation of the numerical model the internal setup in permeable coastal structures is given separate attention. A combination of an analytical prediction of the magnitude of the internal setup and numerical results is used to derive numerically based empirical formulae for the magnitude and the time scale of the internal setup. The formulae include the effects of wave height, wave period, material properties of the coastal structure and the dimensions of the structure. The numerical model is based on the OpenFoam® CFD-toolbox.

1. Introduction

Physical model tests are regularly being used to validate the hydraulic performance of a coastal structure. For instance, physical model tests are often applied to validate the performance of the coastal structure with respect to overtopping, wave forces, wave transmission and armour layer stability; all of these under the influence of irregular waves. The design of the coastal structure, which is eventually tested in the physical model, is partly based on experience from previous projects or existing empirical formulae, though it is not always certain, whether the extrapolation of knowledge is applicable. As an example, the relationship suggested in Foyer and Oumeraci (2012), their Eq. (8), implicitly contains information on the material properties of the core of the coastal structure, thus extrapolation to other material properties would be uncertain. Here, numerical modelling of the coastal structure is useful during the initial design process, because its application can remove many of the uncertainties with respect to the extrapolation of knowledge from other designs. Furthermore, while physical model tests at prototype scale are uncommon (for other purposes than research), it is directly possible to perform the numerical modelling at prototype scale. A practical limitation for physical model testing is the number of available wave flumes and the fact that it can be time consuming to construct a new model or repair damages between tests.

There are many flavours of numerical models that are being used in the coastal zone. It is methods like Boussinesq type equations (Madsen and Schäffer, 1998), nonlinear shallow water equations with a non-hydrostatic solution to the pressure (Ma et al., 2014 and Zijlema and Stelling, 2008) and the more expensive volume of fluid method (VOF), where the free surface can attain arbitrarily complex geometries. The latter modelling framework can handle overturning waves and slamming forces on structures and this modelling approach is adopted in this paper. Consequently, the authors will constrain the following discussion to this method and the term 'numerical modelling' will refer to VOF-type models.

The usage of numerical modelling comes with its own uncertainties and limitations. One of the uncertainties is for instance the limited amount of validation work for the prediction of wave transformation of irregular waves and the interaction between these waves and permeable coastal structure. The validity of the present numerical model to be applied to long time series of irregular (normal) incident waves is one of the key elements of this work. Previously, Torres-Freyermuth et al. (2007) and Losada et al. (2008) used the model COBRAS-UC to investigate the propagation of irregular waves over a cross-shore profile and the interaction with a coastal structure, and Jensen et al. (2014a,b) applied OpenFoam to simulate overtopping and forces on crest elements under the forcing of incident irregular waves. Most other works investigated short time series with regular or solitary waves (Liu et al., 1999; Shen et al., 2004; Zhan et al., 2010 and El Safti and Oumeraci, 2013 to name only a few). A second limitation is the requirement for extensive

* Corresponding author. Tel.: +31 64 69 111 64.
E-mail address: niels.jacobsen@deltares.nl (N.G. Jacobsen).

http://dx.doi.org/10.1016/j.coastaleng.2015.05.004
0378-3839/

computational resources, especially for three dimensional simulations such as the results presented in Higuera et al. (2014b). The computational requirements for two dimensional simulations (one horizontal and one vertical dimension) are considerably smaller and practical experience has shown that it is possible to perform tens of simulations with varying environmental forcing within a matter of days; this obviously depends on the availability of computational resources. With the further development of CPUs, acceleration of numerical methods and the beginning migration to graphic cards (GPUs), the computational efforts for such two dimensional simulations will hardly be an issue in the future.

Several models have been and are being developed for the purpose of wave interaction with permeable, coastal structures. Van Gent et al. (1994) modelled the interaction between waves and a berm breakwater and compared the experimental and numerical values for the pressures inside the structure. The model COBRAS (later named COBRAS-UC and IH2VOF) has been used for a large range of applications: the interaction between waves and coastal structures and basic understanding of the surf zone processes (Liu et al., 1999; Losada et al. (2008) and Ruju et al. (2012)). These models are restricted to two dimensional applications, whereas the models ComFlow (Wellens et al., 2010), IH3VOF (Del Jesus et al., 2012) and OpenFoam (Higuera et al., 2014a,b; Jensen et al., 2014b and Paulsen et al., 2014) can be applied to investigate the interaction between waves and permeable or impermeable structures in three dimensions. Of these models, OpenFoam has most recently been extended with the capability of modelling waves and permeable structures (see below) and OpenFoam seems to attract a growing community for coastal, offshore and maritime engineering, but validation of its usefulness for the modelling of long time series of irregular waves is to the authors' knowledge still lacking.

OpenFoam is distributed under an open-source licence and there are currently two approaches available for the generation and absorption of free surface waves, namely the waves2Foam package, which is based on the use of relaxation zones (Jacobsen et al., 2012), and a method that imposes the velocity field directly at the boundary with a Dirichlet type boundary condition. This latter method applies a mixed Dirichlet/Neumann boundary condition for the indicator function of the volume of fluid field (Higuera et al., 2013). The reflection compensation in the method by Jacobsen et al. (2012) is included over the full length of the relaxation zones, while the reflection compensation is evaluated directly at the boundary in the method described by Higuera et al. (2013). Their method is based on the Sommerfeld transmissive boundary condition that is based on an assumption of shallow water waves. It was shown in Wellens (2012; Chapter 6) that a large amount of wave reflection of an irregular wave field is to be expected from an absorbing boundary condition that is based on the Sommerfeld transmissive boundary condition. Therefore, the waves2Foam method is adopted.

This work will be organised as follows: In Section 2, a description of the numerical model will be given. In Section 3, the focus will be on the validation of the present numerical model for cases with irregular (normal) incident waves with long time series of 500–1000 waves. The results are compared with laboratory scale physical experiments with permeable and impermeable structures. Quantities such as wave reflection, wave damping and wave induced forces will be evaluated. In Section 4, the numerical model will be applied to the coastal engineering problem of internal setup in permeable structures with a sloping front. This will give an example of the combined application of analytical works (presented in Appendix 1) and numerical modelling that leads to numerically based empirical formulae for the magnitude and time scale for the internal setup. The work is completed with conclusions.

2. Model description

In the present work, the Navier–Stokes equations are solved together with a tracking of the free surface with a VOF approach. The effect of the flow resistance due to the presence of a permeable core is included through the Darcy–Forchheimer approximation. In the following section the modelling framework is briefly outlined.

The modelling framework was the open-source computational fluid dynamics toolbox OpenFoam, which is a finite volume approach with a collocated variable arrangement on generally unstructured grids. The OpenFoam version 1.6-ext was used in the present work. At the moment of writing, the most recent OpenFoam version from OpenCFD Ltd. (ESI Group) is version 2.3.1, but the authors have applied version 1.6-ext, because this version enables the use of finite area meshes (Tukovic, 2005) and more robust mesh motion techniques based on a finite element approach (Jasak and Tukovic, 2006). Both of these functionalities are needed for e.g. the modelling of cross-shore morphodynamics (Jacobsen and Fredsøe, 2014) and pipeline scour (Fuhrman et al., 2014). Neither of these functionalities is available in the OpenFoam version 2.3.1 from OpenCFD Ltd. (ESI Group). The authors are not aware of any significant differences between the versions, which could affect the accuracy of the simulations for coastal engineering problems.

2.1. The flow equations

The flow equations consist of the Navier–Stokes equations and the incompressible continuity equation described in terms of the filter velocity:

$$\nabla \cdot \boldsymbol{u} = 0. \tag{2.1}$$

Here, $\boldsymbol{u} = (u, v, w)$ is the vector of the filter velocity in Cartesian coordinates and u, v and w are the velocity components in the three Cartesian coordinates x, y and z. In a permeable structure it holds that $\boldsymbol{u}_p = \boldsymbol{u}/n$, where $\boldsymbol{u}_p$ is the pore velocity and n is the porosity.

The Navier–Stokes equations for flows in a permeable structure were re-analysed in Jensen et al. (2014a), and it was their implementation that was used in the present work. Therefore, the final set of equations is simply stated here:

$$(1+C_m)\frac{\partial}{\partial t}\frac{\rho \boldsymbol{u}}{n} + \frac{1}{n}\nabla \cdot \frac{\rho}{n}\boldsymbol{u}\boldsymbol{u}^T = -\nabla p^* + \boldsymbol{g}\cdot\boldsymbol{x}\nabla\rho + \frac{1}{n}\nabla\cdot(\mu+\mu_t)\nabla\boldsymbol{u} - \boldsymbol{F}_p. \tag{2.2}$$

Here, ρ is the density of the fluid, p^* is the excess pressure, $\boldsymbol{g}$ is the acceleration due to gravity, $\boldsymbol{x} = (x, y, z)$ is the Cartesian coordinate vector, μ is the dynamic molecular viscosity and μ_t is the dynamic turbulent viscosity (see below). The terms C_m and $\boldsymbol{F}_p$ are related to the flow in a porous medium and they will be described in Section 2.2. The relationship between the total and excess pressures is given as:

$$p^* = p - \rho\boldsymbol{g}\cdot\boldsymbol{x}.$$

The free surface is captured using the available VOF-scheme in OpenFoam, and it solves the advection of an indicator function F as follows:

$$\frac{\partial F}{\partial t} + \frac{1}{n}\nabla\cdot\boldsymbol{u}F + \frac{1}{n}\nabla\cdot\boldsymbol{u}_r F(1-F) = 0. \tag{2.3}$$

Here, $\boldsymbol{u}_r$ is a relative velocity (see Berberovic et al. (2009) for details), which aids in retaining a sharp interface. The term $F(1-F)$ vanishes everywhere except at the interface. The indicator function is 1, when the computational cell is filled with water, and 0, when it is filled with air; an intermediate value will be at or close to the interface. The following linear weighting of the fluid properties was adopted:

$$\rho = F\rho_1 + (1-F)\rho_0 \quad \mu = F\mu_1 + (1-F)\mu_0\,. \tag{2.4}$$

Here, the sub-indices 1 and 0 refer to water and air properties, respectively. The correction by the factor $1/n$ in Eq. (2.3) ensures that