©SNAK, 2014

Int. J. Nav. Archit. Ocean Eng. (2014) 6:297~306
http://dx.doi.org/10.2478/IJNAOE-2013-0180
pISSN: 2092-6782, eISSN: 2092-6790

Simulating three dimensional wave run-up over breakwaters covered by antifer units

A. Najafi-Jilani[1], M. Zakiri Niri[1] and Nader Naderi[2]

[1]Civil Engineering, Islamic Azad University, Islamshahr Branch, Iran
[2]Young Researchers and Elite Club, Islamic Azad University, Islamshahr Branch, Iran

ABSTRACT: *The paper presents the numerical analysis of wave run-up over rubble-mound breakwaters covered by antifer units using a technique integrating Computer-Aided Design (CAD) and Computational Fluid Dynamics (CFD) software. Direct application of Navier-Stokes equations within armour blocks, is used to provide a more reliable approach to simulate wave run-up over breakwaters. A well-tested Reynolds-averaged Navier-Stokes (RANS) Volume of Fluid (VOF) code (Flow-3D) was adopted for CFD computations. The computed results were compared with experimental data to check the validity of the model. Numerical results showed that the direct three dimensional (3D) simulation method can deliver accurate results for wave run-up over rubble mound breakwaters. The results showed that the placement pattern of antifer units had a great impact on values of wave run-up so that by changing the placement pattern from regular to double pyramid can reduce the wave run-up by approximately 30%. Analysis was done to investigate the influences of surface roughness, energy dissipation in the pores of the armour layer and reduced wave run-up due to inflow into the armour and stone layer.*

KEY WORDS: Wave run-up; Antifer unit; Numerical simulation; Flow3D.

INTRODUCTION

Wave run-up over coastal structures and breakwaters is one of the most important hydraulic reactions used in crest elevation of designs for coastal structures. The design of rubble mound breakwaters is typically based on empirical formula and physical modeling. One limitation of this approach is that there are many different aspects of wave interaction with a breakwater, such as elevation of the run-up tip and armour stability and these are treated separately. Nowadays the development and evaluation of numerical models to simulate run-up phenomenon is one of the most interesting topics in coastal engineering.

Numerical models are useful tools applied to the early design stages of coastal structures, they are used to help coastal physical modeling in the design tests to measure quantities that are difficult to assess in other modeling techniques such as prototype and scaled down physical models (Altomare et al., 2012). Until recently, numerical models could not predict the complex wave situations that occur in front of breakwaters. So far, wave runup and overtopping on rubble mound breakwaters have been investigated by many researchers (Van Der Meer et al., 2005).

There are two viewpoints for investigating the interaction between fluid and structure: the first supposes that a porous medium has no effect on characteristics and flow pattern and as such a structure's geometry is modeled according to a solid object with a porosity value. In such cases, calculations made out of a simple assumption do not work well in evaluating flows

Corresponding author: *Nader Naderi*, e-mail: *n.naderi@live.com*

with high Reynolds and in cases when a breakwater constitutes large pieces such as concrete armour pieces. In the second viewpoint, porous medium geometry is modeled precisely and in such cases, fluid can move inside a porous solid object (Dentale et al., 2012). The second model obviously requires fabricating a shape's complex geometry and applies more computational cells, but it provides a more accurate model. Dentale et al. (2009) introduced an innovative RANS/VOF procedure by integrating CAD and CFD software to analyze the hydrodynamic aspects of the interactions between breakwaters and waves. Their method produced better results than the traditional approach whereby the flow within the armour is computed with seepage flow approximation (Cavallaro et al., 2012). Latham et al. (2008) used discrete element and combined finite-discrete element methods to model the granular solid skeleton of randomly packed units coupled to a CFD code which resolves the wave dynamics through in interface tracking technique. Latham et al. (2008; 2013) and Xiang et al. (2012) demonstrated that FEMDEM provides excellent shape representation and deformability for static and dynamic problems for faceted and angular concrete units and rock blocks used in armour layers. Their method also provides a powerful tool for examining stress chains within granular packs of armour units.

This present research work investigates the wave run-up on breakwaters covered by antifer units via Flow-3D and AutoCAD software, and is supported by the second mentioned viewpoint that incorporates an accurate geometric model of the breakwater. AutoCAD is utilized to construct different parts of the breakwater's geometry such as core, toe and armor layers. In Fig. 1, armour and other used stones into the stone layer and the breakwater's toe are also shown.

RANS type RNG model is used for the turbulence modeling. The numerical results obtained by Bakhtyar et al. (2010) revealed that the RNG turbulence model yielded better predictions of nearshore zone hydrodynamics, although the k-ε model gave satisfactory predictions.

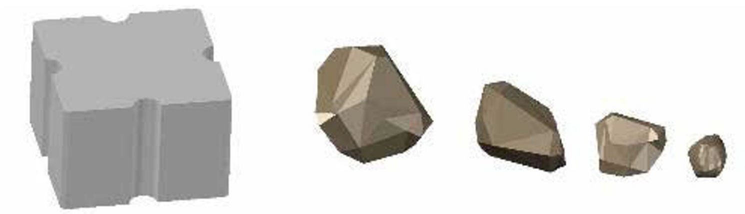

Fig. 1 Virtual 3D models of stones and antifer.

CFD model

Flow-3D is the computed fluids dynamic model in this research work. The software is capable of considering different boundary conditions of fluid and to solve the common interface of fluid with fluid and fluid with air. Proposed model is utilized in different applications of hydraulic engineering and coasts such as flow and erosion around hydraulic structures and transmission of waves near the beach. Flow-3D solves three-dimensional Navier-Stokes equations and continuity equation simultaneously. The flow is described by the general Navier-Stokes equations (Dentale et al., 2012):

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

$$\frac{\partial u_i}{\partial t} + u_j \frac{\partial u_j}{\partial x_j} = \frac{1}{\rho}\frac{\partial \rho}{\partial x_i} + \nu \frac{\partial^2 u_i}{\partial x_j \partial x_j} + g_j \tag{2}$$

where ν is the molecular viscosity, u_i is the ith component of the instantaneous velocity in the pores, p the instantaneous effective pressure and g_i the ith component of the gravitational force.

Flow-3D is a non-hydrostatic model. Non-hydrostatic models do not apply simple assumptions into momentum equations, so these must be solved repeatedly. Flow-3D uses a method that considers 'volume of fluid' (VOF) to study the common interface of fluid with air and fluid with fluid. This method records levels of fluid volume for each cell. This volume is then used to determine comparisons with evaluations for cell adjutant volumes for ant, slope, position and curvature of the fluid into