©SNAK, 2014

Int. J. Nav. Archit. Ocean Eng. (2014) 6:800~812
http://dx.doi.org/10.2478/IJNAOE-2013-0214
pISSN: 2092-6782, eISSN: 2092-6790

# Numerical wave interaction with tetrapods breakwater

Fabio Dentale[1,2], Giovanna Donnarumma[1] and Eugenio Pugliese Carratelli[1,2]

[1]*University of Salerno-MEDUS (Maritime Engineering Division University of Salerno)*
*Department of Civil Engineering - Via Giovanni Paolo II, 132 - Fisciano (SA), Italy*
[2]*CUGRI (University Consortium for Research on Major Hazards) -*
*Via Giovanni Paolo II, 132 - Fisciano (SA), Italy*

**ABSTRACT:** *The paper provides some results of a new procedure to analyze the hydrodynamic aspects of the interactions between maritime emerged breakwaters and waves by integrating CAD and CFD. The structure is modeled in the numerical domain by overlapping individual three-dimensional elements (Tetrapods), very much like the real world or physical laboratory testing. Flow of the fluid within the interstices among concrete blocks is evaluated by integrating the RANS equations. The aim is to investigate the reliability of this approach as a design tool. Therefore, for the results' validation, the numerical run-up and reflection effects on virtual breakwater were compared with some empirical formulae and some similar laboratory tests. Here are presented the results of a first simple validation procedure. The validation shows that, at present, this innovative approach can be used in the breakwater design phase for comparison between several design solutions with a significant minor cost.*

***KEY WORDS:*** Volume of Fluid (VOF); Wave; Run up; Reflection; Rubble mound; Numerical simulations; Tetrapod; Flow 3D®; RANS equations.

## INTRODUCTION

Coastal structures, and specially so rock mound breakwaters, are normally designed by using well proven formulas and by laboratory scale tests.

Recently 2D and 3D numerical simulation of Navier Stokes equation has been developed to the point that it can now be used as an affordable design tool to substitute or supplement tank experiments. The references on this topic are far too many to be examined in detail here, however it may be useful to recall some interesting examples of how these issues have been addressed both physically and numerically (developed by Lin and Liu, 1998; Giarrusso et al., 2003; Losada et al., 2008).

Mounds of rock or concrete blocks however still seem to defy the computational possibilities, due to the geometrical and hydrodynamical complexity of the problem. Water flows through the paths of the interstices within the blocks featuring strongly non stationary flow, free boundaries, turbulence, interaction with solid transport and complex geometry (Koutandos et al., 2006a; 2006b).

The currently used approach assumes that within the rubble mound the flow can be treated by using a classical "porous

Corresponding author: *Fabio Dentale*, e-mail: *fdentale@unisa.it*

media" methodology, i.e. by using, within the rubble mound, the equations that treat the filtration motion (Darcy or Forchheimer, if the head loss is linear or quadratic respectively).

In practice, an additional term is added to the equations to reproduce the interactions between the fluid and the inner flow paths using homogeneous coefficients for the entire filtration domain.

Such an approach was reported in Hsu et al. (2002), later implemented in the COBRAS numerical code and finally perfected by Lara et al. (2006).

The results obtained through these types of modeling, while certainly more reliable compared to the waterproof block model, present a number of drawbacks. First of all, this approach overlooks the convective aspects of the flow and the structure of turbulence; it is heavily reliant on of the numerical parameters of the filtration equations and therefore it requires a careful empirical calibration.

Only recently serious attempts have been made to model the detailed hydrodynamics of block mound structures on the basis of their real geometry by using advanced digital techniques: by using a fine computational grid, an adequate number of computational nodes is located within the interstices so that a complete solution of the full hydrodynamic equations is carried out, thus including convective effects and possibly also resolving the turbulence structure. All these aspects cannot be taken into account with the classical porous media approach, inadequate in such kind of situations.

Pioneering work with full simulation of such Flow Within the Armour Units (FWAU) method was carried out by using RANS-VOF, (Cavallaro et al., 2012; Dentale et al., 2012; 2013; 2014); Smoothed Particle Hydrodynamics (SPH) was applied to this problem by Altomare et al. (2012), while a somewhat similar approach involving CFD techniques in the interstices and numerical solid mechanics in the block themselves, is being attempted by Xiang et al. (2012).

The aim of the present work is to introduce a validation of the procedure against classical empirical formulas and physical tests for a structure with rubble mound in Tetrapods and to show how it is already a useful design tool in some complex configurations.

## PROCEDURE

The innovative approach, should in principle be three-dimensional since the geometrical structure of the interstices among the blocks has inherently a very complex spatial structure; some successful attempts have indeed been made by the Authors in (Dentale et al., 2009) to develop equivalent 2D schemes, but they have not been followed in the present work .

Numerical reconstructions of the breakwater are thus produced by using a CAD software system for modeling 3D geometries; a data base of artificial blocks such as the cube, the modified cube, the Tetrapod, the Seabees (Brown and Dentale, 2013), the Accropode™ and the Xbloc®, has preliminarily been produced, while also natural rocks can be reproduced either by using spheres of various diameters or by randomly shaped blocks.

Breakwaters, both submerged and emerged, are numerically reconstructed by overlapping individual blocks under the conditions of gravity, collision and friction, according to the real geometry, very much like in the case of real constructions or laboratory test model.

In this case a classical breakwater is reconstructed by a CAD software with the following scheme:

- A waterproof core;
- A filter layer in natural stones;
- A toe protection in natural stones;
- An armour layer in Tetrapod reconstructed in respect of collision, gravity and interlocking forces.

Once the breakwaters geometry is defined (Fig. 1), its geometric configuration is imported into the CFD system.

FLOW-3D® (Flow Science Inc.) was used for all calculations, like many other CFD systems employed for similar tasks, FLOW-3D® is based on the Reynolds Averaged Navier-Stokes (RANS) equations combined with the Volume of Fluid (VOF) method to apply the proper dynamic boundary conditions and to track the location of the fluid surfaces (Hirt and Nichols, 1981).