IOP Conf. Series: Materials Science and Engineering **400** (2018) 082003 doi:10.1088/1757-899X/400/8/082003

Nearshore wave processes numerical study for a better prediction of hydrodynamic loads on coastal structures

I A Anton, A Scupi and D Dinu
Constanta Maritime University, Faculty of Naval Electromechanics, 104 Mircea cel Batran, 900663 Constanta, Romania

E-mail: iulia.anton@cmu-edu.eu

Abstract. Taking into account the sensible approach in the coastal zones, in these types of areas were found different ways of understanding and description of the phenomena occurring during the water wave propagation. The waves hit the zone with a force that can affect the shoreline. Coastal protection measures were implemented, which include the hydrodynamic structures. The hydrodynamic process plays an important role in coastal area which involves the need for knowledge on the prediction of hydrodynamic loading required up to a very detailed level, such as maximum or minimum pressures. In this paper, we will present a study of wave fields that propagates in a simplified, constant-slope coastal model and relations between waves and hydrodynamics structures. The processes of wave fields include dissipative mechanisms such as depth-induced wave breaking, bottom friction and the interaction between wave field and coastal dike. The numerical models were constructed using ANSYS-Fluent program for modelling fluid flow and to solve the unsteady, two dimensional Navier-Stokes equations of incompressible fluid. Water waves are generated in two-phase fluid domain, where Volume of Fluid Model is used for generation of the two-phase flow model - the water-air interaction.

1.Introduction

Economic activity grows more and more every day. At the moment, a new area open to research is the study of fixed or floating marine structures with different destinations (houses, research laboratories, power generating structures, etc.). To study the wave-related risks on these types of structures, the researchers need to understand the phenomena that occur in the wave-structure interaction and the distribution of the various parameters (velocity, pressure, etc.) following wave breaks. Of course, experiments are also done in the laboratory, in a wave tank, involving various tests on existing models. However, once most discussions will be given by the results of computerized fluid dynamics software, in which we can simulate various phenomena in an inexpensive and fast way [1-4].

Computerized Dynamics Software is based on numerical simulations of non-linear waves using a finite differential method, the finite element method.

Fluent software package, used to study wave interaction with a marine structure, is ANSYS, 2009 that used a finite volume method to solve the Reynolds Averaged Navier-Stokes (RANSE) equations explaining turbulence and viscosity [5].

The equations behind this model are the Navier-Stokes equations with kinematic dynamics and the VOF (Volume of Fluid) method used to simulate the presence of fluid in each cell. Mass and moment consistency equations are used for initial flow without air-wave interference.

IOP Conf. Series: Materials Science and Engineering **400** (2018) 082003 doi:10.1088/1757-899X/400/8/082003

The VOF method captures interfaces in which the location of the interface is captured by tracking the volumetric fraction of each computational cell on the network relative to one of the fluid phases: cells that have a zero volume or unit size fraction do not contain an interface, while those that have a fractional value contain. Thus the free surface is rebuilt. However, a very fine mesh for the reconstruction of the liquid surface is required [4].

2. The governing equations

The finite volume method that is used in ANSYS-Fluent divides the area into small areas and discretizes the governing equations in order to solve them iteratively over each sub-regions [4]. Therefore, an approximation of the value of each variable at points throughout the domain is achieved.

The governing equations that we used are the mass continuity equation:

$$\frac{\partial p}{\partial t}+\frac{\partial \rho u_1}{\partial x}+\frac{\partial \rho u_2}{\partial y}=0\ , \tag{1}$$

and the Navier-Stokes equations:

$$\rho\left(\frac{\partial u_1}{\partial t}+u\frac{\partial u_1}{\partial x}+v\frac{\partial u_1}{\partial y}\right)=-\frac{\partial p}{\partial x}+2\mu\frac{\partial^2 u_1}{\partial x^2}+\frac{\partial}{\partial x}\left(\mu\left(\frac{\partial u_1}{\partial y}+\frac{\partial u_2}{\partial x}\right)\right)+F_1, \tag{2}$$

$$\rho\left(\frac{\partial u_2}{\partial t}+u\frac{\partial u_2}{\partial x}+v\frac{\partial u_2}{\partial y}\right)=-\frac{\partial p}{\partial y}+2\mu\frac{\partial^2 u_2}{\partial y^2}+\frac{\partial}{\partial x}\left(\mu\left(\frac{\partial u_1}{\partial y}+\frac{\partial u_2}{\partial x}\right)\right)-\rho g+F_2, \tag{3}$$

where t is time, x is the horizontal distance from the wave maker, y is the vertical height and increases with depth, u_1 is the horizontal flow velocity, u_2 is the vertical flow velocity, F_1 is the horizontal force on the fluid, F_2 is the vertical force on the fluid, p is pressure and μ is viscosity [3].

To calculate the two-phase transportation (air and water) we have used the volume fraction method. The volume fraction method relies on the fact that two or more fluids are not interpenetrating. For each additional phase that is added to the model, a variable is introduced: the volume fraction of the phase in the computational cell. In each control volume, the volume fraction of all phases sum to unity. Thus, the variables and properties in any given cell are either purely representative of one of the phases, or representative of a mixture of the phases, depending upon the volume fraction values [3].

$$\frac{1}{\rho_q}\left[\frac{\partial}{\partial t}\left(\alpha_q\rho_q\right)+\nabla\cdot\left(\alpha_q\rho_q\right)=S_{\alpha_q}+\sum_{p=1}^{n}\left(\dot{m}_{pq}-\dot{m}_{pq}\right)\right], \tag{4}$$

where: $\dot{m}_{qp}$ is the mass transfer from phase q to phase p and $\dot{m}_{pq}$ is the mass transfer from phase p to phase q; α_q is the volume fraction of the phase q and S_{α_q} is a specific constant [3].

For the waves theory we use Stokes fifth order which is appropriate for intermediate and shallow water, given by the equations for wave's profiles:

$$\zeta(X,t)=\frac{1}{k}\sum_{i=1}^{5}\sum_{j=1}^{i}b_{ij}\xi^i\cos(j\alpha)\,, \tag{5}$$

where α is a variable, ξ is the wave steepness:

$$\xi=\frac{kH}{2}, \tag{6}$$

where h is wave height, k is the wave number

$$k=\frac{2\pi}{\lambda}, \tag{7}$$

where λ is the wave length.

3.Numerical Model

One way to significantly reduce multiple simulation efforts is to use ANSYS FLUENT mesh functions to describe water-air interference by efficiently solving the defined liquid surface using numerical diffusion. The other surfaces were shaped as walls [4].

The flow system is laminar and isothermal, water and air being considered fluid. The domain in drawn in 2D and consists in 2 parts, an upper part (fluid domain: water and air) and a downer part (the soil). The domain is 2000m long and 50 high. The soil increases from 0 m to 26 m in about 1500m. The shoreline was imposed at 25m high from domain, figure 1.

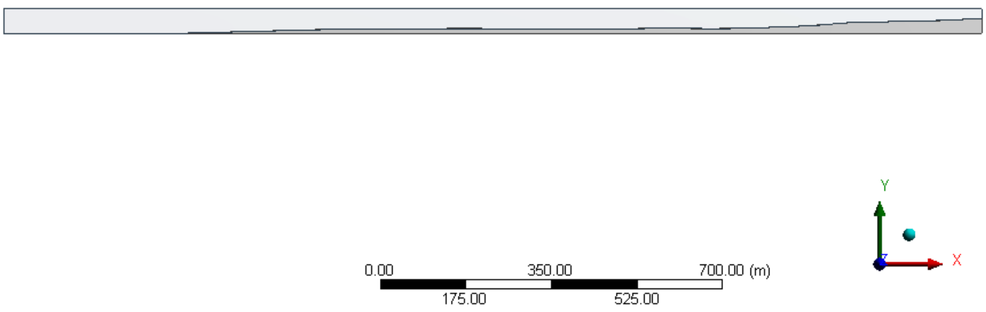

Figure 1. The shoreline represented in 2D.

The mesh (figure 2) consists in over 100000 cells with almost all cells quality close to 1. This was imposed to obtain accurate results when dealing with 2 phase fluid flow (air and water) and fluid-solid interaction.

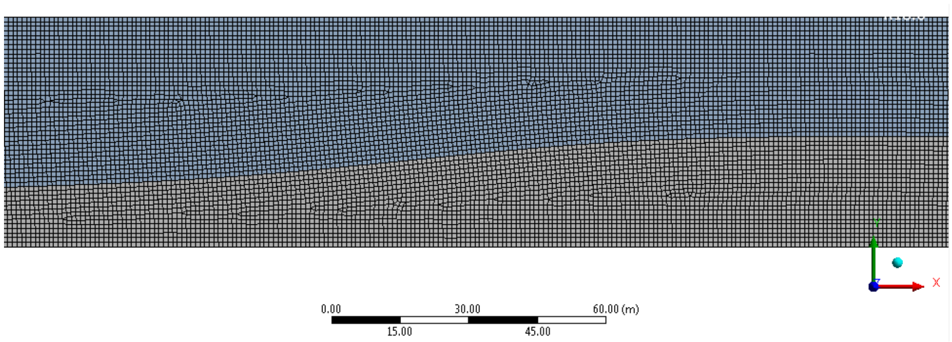

Figure 2. The mesh of shoreline.

The boundary conditions were considered as:

- 2 phase (air and water) flow;
- Wave characteristics: 1 m height and 20 m long and 4 m/s celerity.
- K-epsilon turbulent model and 5th order Stokes model for wave propagation.

4. Results and discutions

We simulated two situation: the first one wave propagation close to the shore with no wave breaker and a second one whit and wave breaker.

In figure 3a and 3b are presented density and pressure distributions for the fluid domain and it can be observed that far away from the shore the waves reproduces symmetrically.

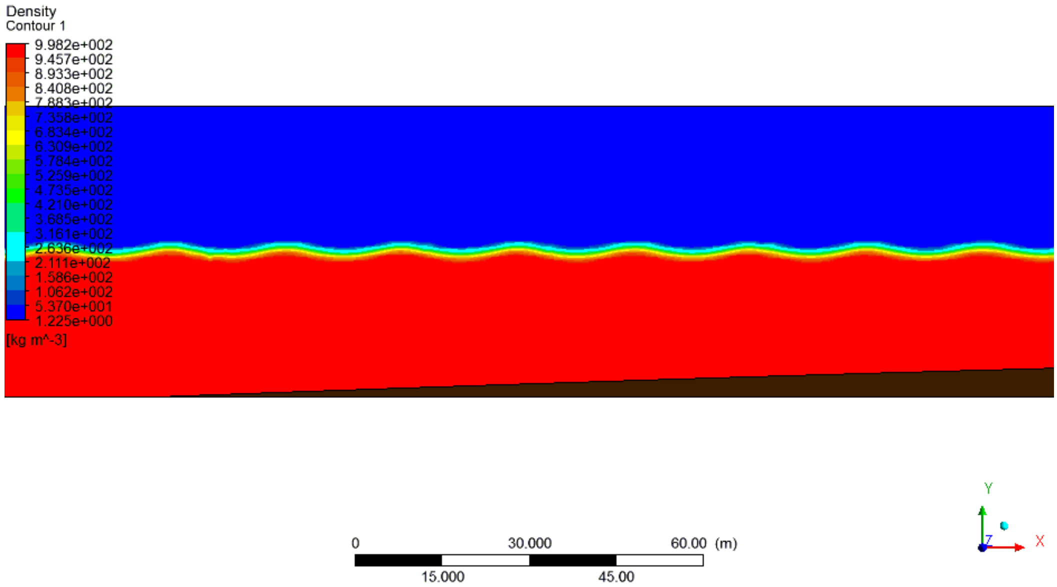

Figure 3a. Density distribution for the fluid domain.

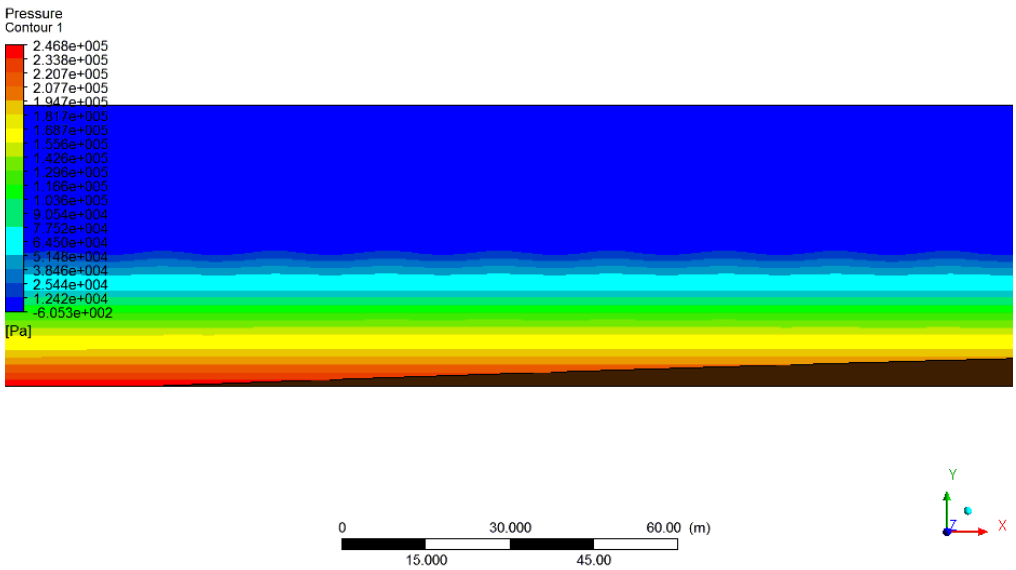

Figure 3b. Pressure distribution for the fluid domain.

While close to the shore (figure 3c and 3d), the waves flatten and their form is similar to an ellipse [2]. In figures 3c and 3d the soil was made transparent and a vertical reference line was created at the intersection between shoreline and free water surface so we can easily observe wave propagation on shoreline.

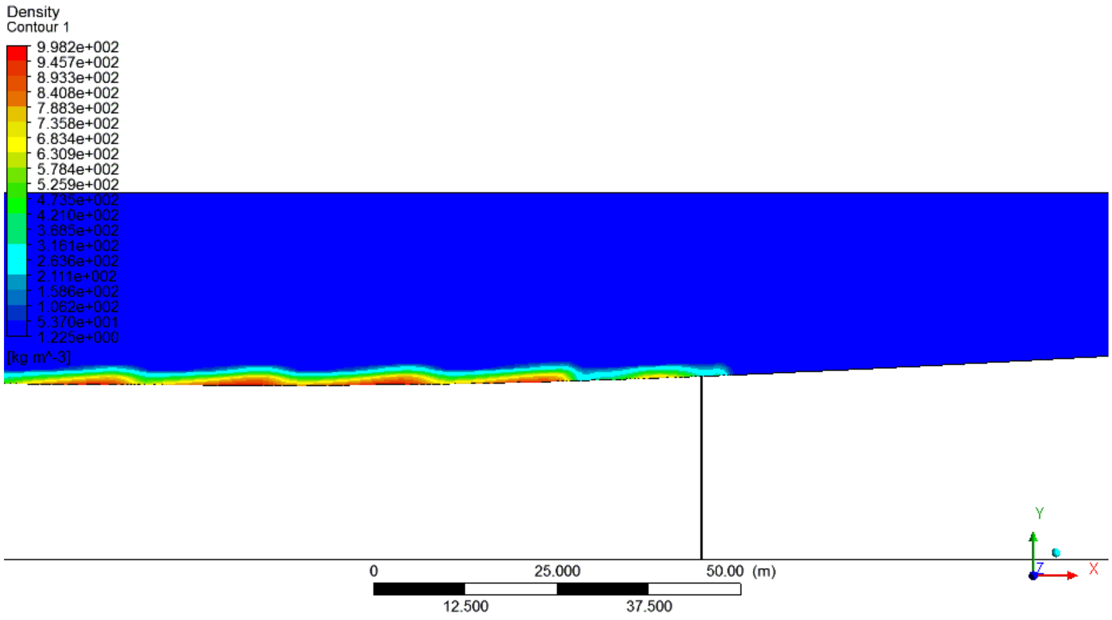

Figure 3c. The density distribution in breaking waves phenomena.

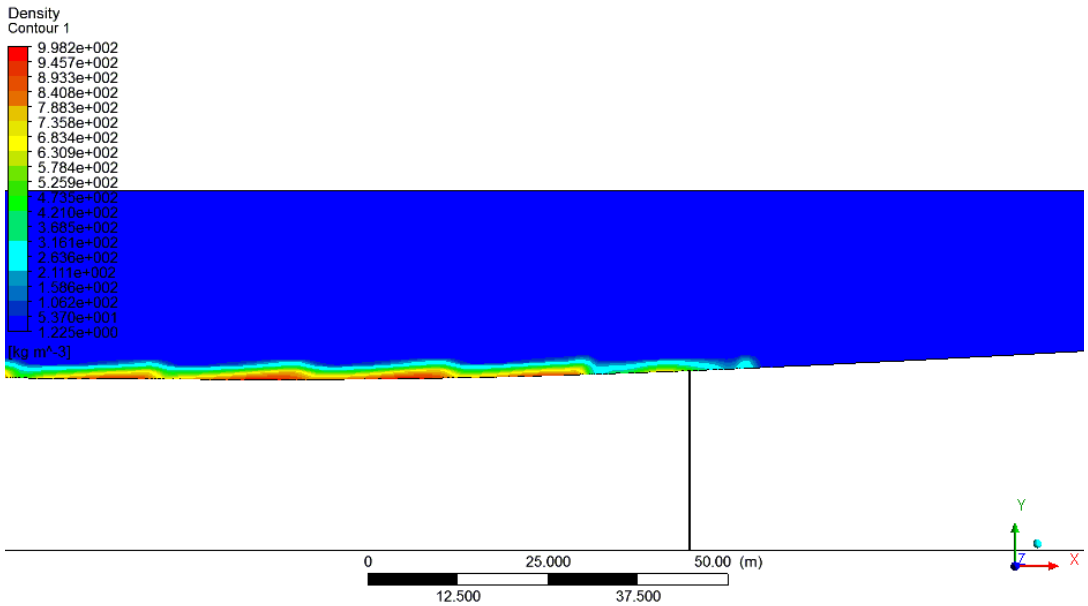

Figure 3d. The density distribution in breaking waves phenomena during time steps.

If waves encounter an obstacle such as a breakwater, the density distribution changes so that part of the wave energy is dissipated, thus protecting the shoreline. The wave breaker height, figure 4, reaches the free water surface and we can observe that the wave changes its characteristics, wave height is smaller and wave length is longer.

The program allows us to plot pressure variation on the wave breaker which is very helpful when determining the shape and the dimension on the wave breaker, as we see in figure 5.

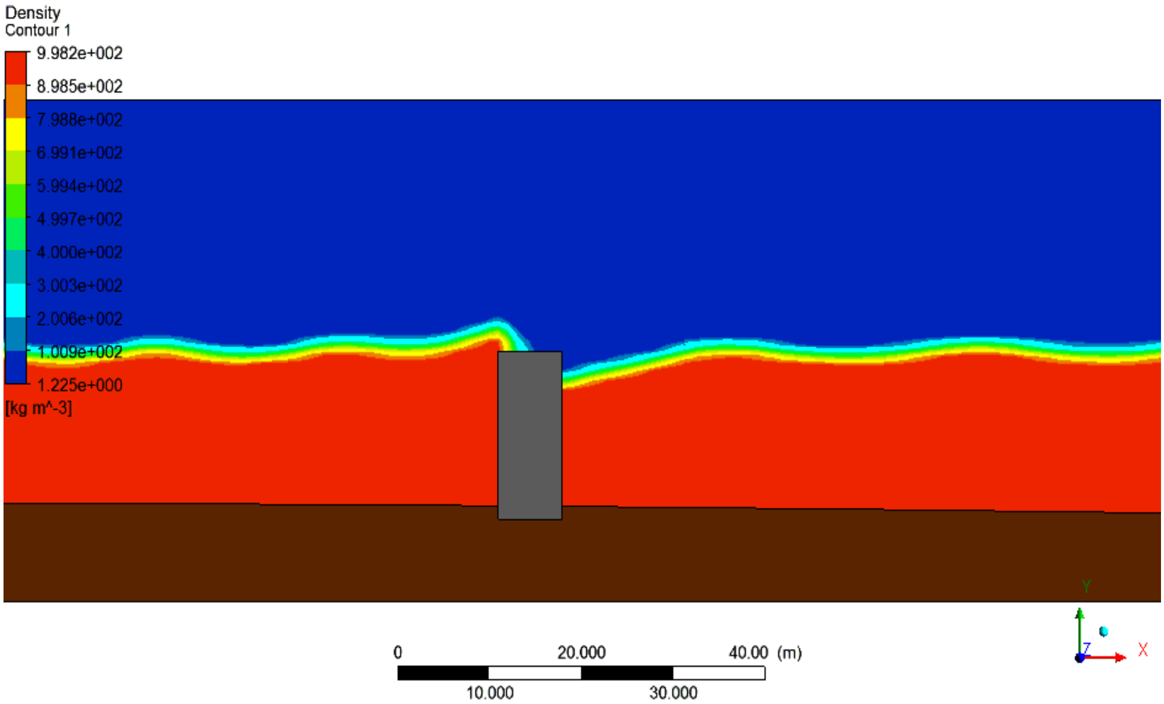

Figure 4. The density distribution between breakwaters and fluid.

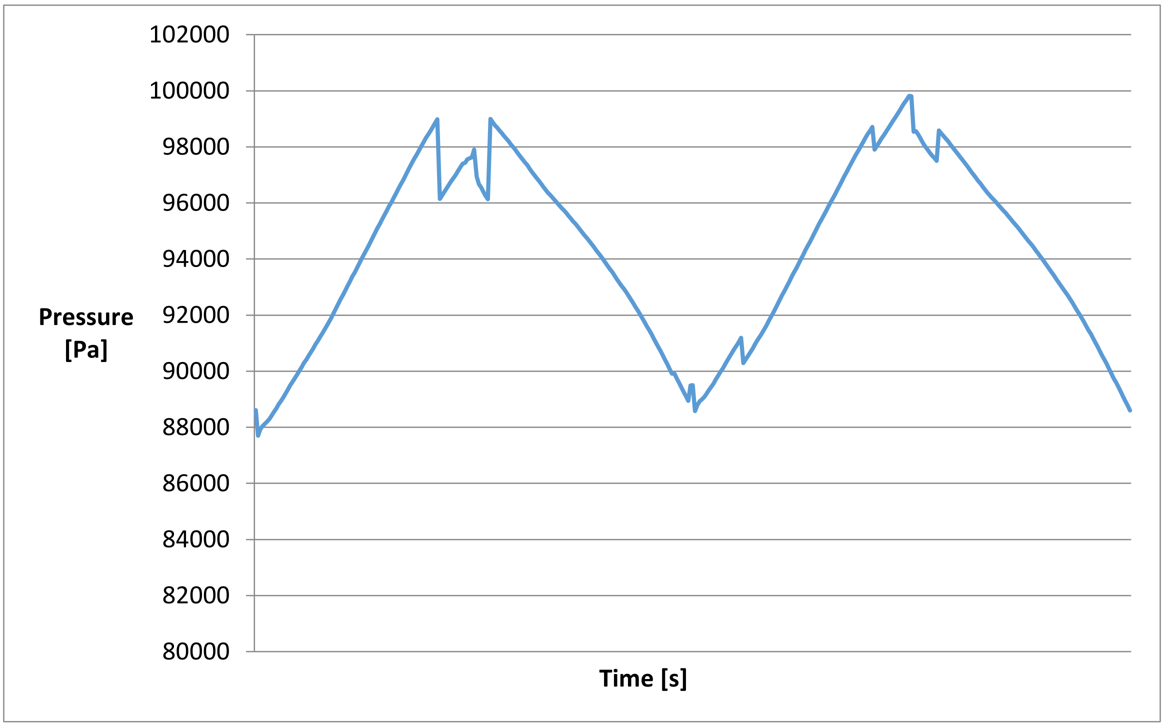

Figure 5. Pressure variation on the wave breaker.

Pressure on the wave breaker was computed as an area weighted average on the all surface and it has an sinusoidal shape appropiate with the sinusoidal wave propagation.

5. Conclusion

The simulations proposed in this work for the analysis of wave breaking phenomena, based on the Volume of Fluid Model method, have shown that the ANSYS Fluent model significantly reduces the numerical computational effort if correct boundary conditions used. Also it gives a good insight to estimate the way the wave changes its characteristics pressure values on the wave breaker, necessary in future researches to choose the correct shape, dimensions for one or more wave breakers.

From the distribution of pressures and densities in a normal regime, without an obstacle in the way of the waves, it was observed that the phenomenon of wave breakage occurs at low depths, when the ridge of the waves reaches the bottom of the sea. In the case of a breakwater, there is a significant change from the normal regime, the distribution being much clearer, which explains the energy dissipation as a result of the impact.

Also this model can explain the effects of air interaction with water surface and wave formation. Energy dissipation occurs when the waves break, the air bubbles trapped inside the water are not negligible. At the time of waves breaking the pressure becomes much lower than the hydrostatic pressure due to the significant movement of the water in the vertical direction. Thus, the velocity of the water particle on the wave ridge is much higher than the speed of the waves in the area.

6. References

[1] Anton I A, Dinu D 2017 Wave Simulation with different type of coast potection structure – a comparative approach *International Journal of Environmental Science* **2** 171-176
[2] Dinu D 1999 *Hydraulics and hydraulic machines* (Constanta: Sigma) p 118-119
[3] Raval A 2008 *Numerical simulation of water waves using Navier-Stokes equations* (PhD thesis, University of Leeds) p 28-61
[4] Scupi A A, Dumitru D 2015 *Fluid Mechanics – Numerical Approach* (Constanta: Nautica Publisher)
[5] ANSYS 2017 Fluent User Guide