23rd ABCM International Congress of Mechanical Engineering
December 6-11, 2015, Rio de Janeiro, RJ, Brazil

OPTIMIZATION OF A VANE-TYPE CARGO PROTECTION USING A CFD APPROACH

Daniel Freitas Coelho
José Manoel Freire Guimarães
Bruno Alberto Cardoso Pignatário
Danilo de Souza Braga
Newton Sure Soeiro
Universidade Federal do Pará, Grupo de Vibrações e Acustica. Rua Augusto Corrêa, 1. Guamá. Belém, Brazil
daniel.coelho@itec.ufpa.br
jose.guimaraes@itec.ufpa.br
bruno.pignatario@itec.ufpa.br
dbraga@ufpa.br
nsoeiro@ufpa.br

Abstract. *When a ship travels or is moored in rough weather, its bow may become immersed in water and a solid and compact mass of water will wash along the deck, which is called green water effect. This has been noted as one of the main causes of damage to ship superstructures, such as: deck plating, hatches and cargo. If it is not necessary to keep an installation on the deck completely dry, it is possible to use breakwaters in front of it. The vane-type breakwater is a good alternative to prevent the run-up of water and reduce the impact load with a lighter structure. Therefore, an optimization of a vane-type breakwater using the dam break method on a free surface unsteady CFD approach, analyzing both the loads on the vane structure and on the critical cargo, was performed. Initially, two vane profiles were investigated: plates and 4-digit cambered NACA airfoils. Since, a NACA profile has a reduction of 56% on the load on the squared wall and only an increase of 10% on the breakwater, compared with a plate profile, the NACA airfoils were chosen to do a parametric optimization using ANSYS' screening method, which will be discussed later.*

Keywords: *optimization, green water effect, vane-type breakwater, free surface, unsteady CFD*

1. INTRODUCTION

In marine and offshore industry, problems that involves highly nonlinear steady and unsteady free surface flow are often present, since ships and offshore structures can suffer serious damage from water impact. With the development of computational resources to solve fluid dynamics equations, it turned possible to analyze real problems with greater accuracy and robustness.

In order to investigate the dynamic loads on an offshore structure, the green water effect must be considered since it is the phenomenon that causes the most damage to ship superstructures, such as deck plating, hatches and cargo. In Norway, waves ranging from 8 m to 12.5 m were recorded and several damages to windows and walkways were taking (Ersdal and Kvitrud, 2000).

Since the knowledge about complex physics of green water flow onto the deck is limited, this phenomenon is often studied using the dam breaking approach. This type of simulation consists on the evaluation of water impact due to a water column positioned ahead of the structure. Other studies (Buchner, 1999; Buchner, 2002; Silva and Rossi, 2012) have proven that the dam breaking model provides a good simplified model of the flow from the green water.

There are several ways to solve the green water problem such as: optimize bow shape; increase freeboard height, or use protecting breakwaters in front of critical structures and equipment (Marin, 2003). If it is not necessary to keep an installation on the deck completely dry, it is possible to use breakwaters in front of it. The vane-type breakwater is a good alternative to prevent the run-up of water and reduce the impact load. This method is more efficient, easier to support on the deck with a lighter structure (Marin, 2003).

This work aims the optimization of the vane-type breakwater configuration using the dam breaking method. Two types of vane profiles were used: plates and 4-digit cambered NACA airfoils. In addition, two configurations of breakwater were considered. The main parameter used to choose the best breakwater were the loads on the vane structure and on the critical cargo. The CFD approach used to solve the dam breaking simulation represents only a part of the vessel deck and a cubic body represents the offshore structures that can suffer from green water effect. It was performed a 3D unsteady free surface simulation for three different water heights (13, 10 and 7 m). All procedures are described further later.

2. GREEN WATER MODELING

Generally speaking, the numerical tools usually have more difficulties to represent quartering and beam sea conditions. For such conditions, a general recommendation is to use the dam break theory as a start point for velocity and loads calculation. With respect to the use of CFD, a lot of research is being done on the detailed simulation of green water events. The CFD approach usually has two possibilities: either a local domain, where only part of the vessel deck is represented or the full simulation of the wave and hull interaction. The second approach is more recent, especially for complex geometries, since it usually implies in huge computational costs. Another possible alternative is to develop a coupling strategy between potential methods for the far field waves and a CFD solver restricted to a region surrounding the deck (Silva and Rossi, 2012).

For this work, it was chosen the simplest method, considering only part of the vessel deck, since results from other authors have proven that this simplification does not imply in considerable errors. The green water phenomenon can be seen on Fig. 1 on the left, where it shows the real problem and on the right, the simplification adopted for the dam break theory. The blue arrows on Fig. 1 (right) denote the interface between the control volume and the atmosphere where pressure is equal to 1 atm. It is also considered that water is displaced in three different heights (13, 10 and 7 m) and the shape of the wave is described by lengths H, d and L according to the methodology adopted by Buchner (2002). The superstructure on the deck is considered as a parallelepiped body to simplify the analysis. The lengths of the deck are shown on Fig. 1 as well.

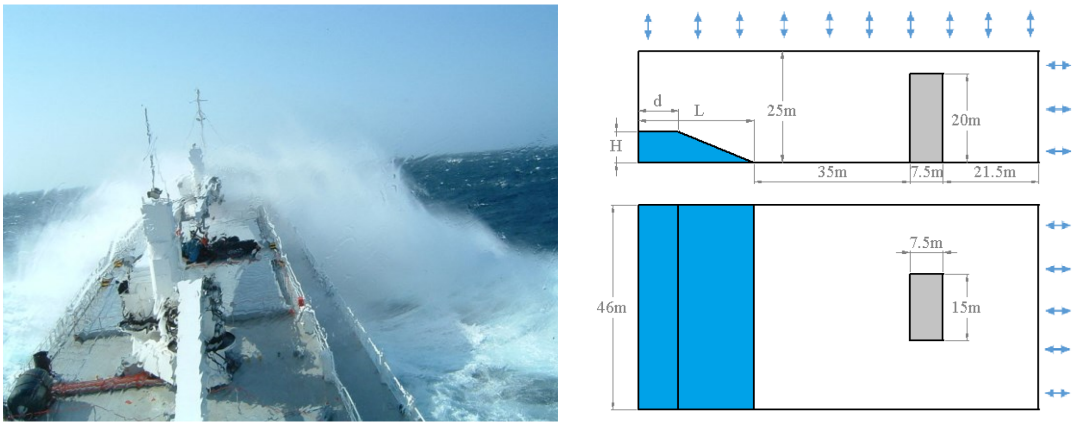

Figure 1. Real green water effect phenomenon and physical model of the problem.

3. VANE-TYPE BREAKWATER

Two types of breakwater configurations were used, plates and NACA airfoils. There are seven parameters that describes the breakwater with NACA profile; they are maximum thickness (t), and position of maximum thickness, maximum camber (m), and position of maximum camber (p), chord (c), angle of attack (α) and distance between foils (Δx). The breakwater height is considered constant as the position of maximum thickness, in order to reduce the number of variables. The configurations and the breakwater profile parameters can be seen in Fig. 2. The breakwaters are placed in front of the parallelepiped body and each configuration is compared in terms of the loads that act both on the superstructure and on the breakwater. Due to ease of using plates against airfoils, it was adopted that if the differences between the loads in two cases are small, it is preferable to choose a plate profile.

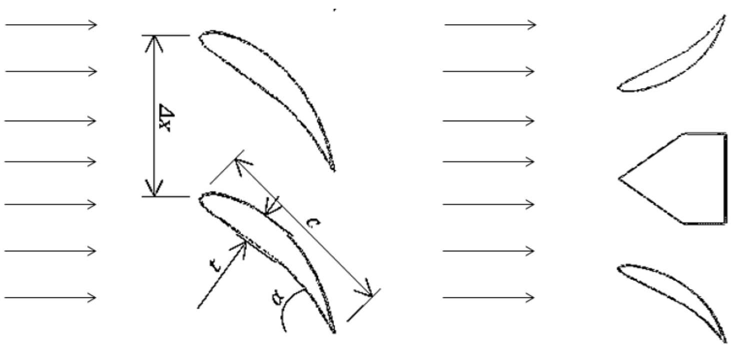

Figure 2. Configurations of the breakwaters.

4. MATHEMATICAL AND NUMERICAL MODELING

4.1 Governing equations of fluid dynamics

The governing equations for the CFD calculations are integrated over each control volume, such that relevant quantities (mass, momentum, energy etc.) are conserved in a discrete sense for each control volume.

In Reynolds averaging, the solution variables in the instantaneous (exact) Navier-Stokes equations are decomposed into the mean (time-averaged) and fluctuating components. Substituting expressions of this form for the flow variables into the instantaneous continuity and momentum equations and taking a time average yields the ensemble-averaged momentum equations (ANSYS Inc., 2006). Considering an incompressible flow, the continuity and momentum equations can be written in Cartesian tensor form, respectively as:

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

$$\frac{\partial \overline{u_i}}{\partial t} + \frac{\partial \overline{u_i u_j}}{\partial x_j} = \frac{1}{\rho}\frac{\partial}{\partial x_j}\left(-\bar{P}\delta_{ij} + \mu\frac{\partial \overline{u_i}}{\partial x_j} - \rho\overline{u'_i u'_j}\right) \tag{2}$$

where $\overline{u_i}(\mathrm{i} = 1,2,3)$ and $\overline{u_j}(\mathrm{j} = 1,2,3)$ are the components of averaged velocity, $\bar{P}$ is the mean pressure, δ_{ij} is the Kronecker delta, ρ is the fluid density, μ is the fluid dynamic viscosity and the term $-\rho\overline{u'_i u'_j}$ is the advection of momentum or Reynolds stress, which results in the spread of momentum by turbulence.

Since there are no explicit governing differential equations for the additional unknowns due to the Reynolds stress term, the problem is indeterminate. One needs to close the problem to obtain a solution. Therefore, the turbulence modeling tries to represent the Reynolds stresses in terms of the time-averaged velocity components and close the problem.

Reynolds stress term can be decomposed, by the Boussinesq approximation, into two parts; an isotropic pressure part termed the turbulence pressure, and an anisotropic part consisting of the turbulent viscosity, as showed respectively on Eq. 3. Where k is the kinect energy of turbulent fluctuations and μ_t is the eddy viscosity.

$$-\rho\overline{u'_i u'_j} = -\frac{2\rho}{3}k\delta_{ij} + \mu_t\left(\frac{\partial \overline{u_i}}{\partial x_j} + \frac{\partial \overline{u_j}}{\partial x_i}\right) \tag{3}$$

It can be seen that to close the problem, it is still needed additional equations to define k and μ_t and so, obtain the Reynolds stress. Therefore, multiple turbulence models were created to calculate these parameters. The turbulence model chosen to this work will be discussed further later.

4.2 Turbulence model

The turbulence is taken into account with a two-equation k–ε model. The main reason to choose this model is because the results obtained seems to be more suitable with experimental data in terms of force prediction for loads on ships and on offshore structures. Turbulence models were also discussed in other studies (Shen, 2003; Yeganeh-Bakhtiary, 2010; Ryu et al, 2007; Hirdaris et al, 2014). The two partial differential equations governing the turbulent kinetic energy k and the turbulence eddy dissipation ε then reads:

$$\rho\left[\frac{\partial k}{\partial t} + \frac{\partial}{\partial x_j}(ku_j)\right] = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right] + P_k - \rho\varepsilon + P_{kb} \tag{4}$$

$$\rho\left[\frac{\partial \varepsilon}{\partial t} + \frac{\partial}{\partial x_j}(\varepsilon u_j)\right] = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_\varepsilon}\right)\frac{\partial \varepsilon}{\partial x_j}\right] + \frac{\varepsilon}{k}(C_{\varepsilon 1}P_k - C_{\varepsilon 2}\rho\varepsilon + C_{\varepsilon 1}P_{\varepsilon b}) \tag{5}$$

where $C_{\varepsilon 1}$, $C_{\varepsilon 2}$, σ_k and σ_ε are constants standardly equal to 1.45, 1.9, 1.0 and 1.3, respectively, P_k is the turbulence production due to viscous forces and P_{kb} and $P_{\varepsilon b}$ represent the influence of buoyancy forces.

Finally, the eddy viscosity for the k–ε turbulence model can be calculated by Eq. 6, where C_μ is a constant, standardly equal to 0.09.

$$\mu_t = C_\mu \rho \frac{k^2}{\varepsilon} \tag{6}$$

4.3 Free surface problem

Free surface flow refers to a multiphase flow situation where the phases are separated by a distinct interface. Examples of free surface flows include open channel flow, flow around ship hulls, tank filling problems, Pelton turbines, and many others (ANSYS Inc., 2006). In homogeneous multiphase flow, a common flow field is shared by all fluids, as well as other relevant fields as temperature and turbulence.

For the calculation of the multiphasic model, it is assumed that each control volume contains just one phase of fluid (or interface between phases) and one set of momentum equations is solved for each fluid. To calculate the volume fraction, a step function ∈ is created in order to calculate the percentage of a fluid inside a cell and it is defined as:

$$\in^{cell} = \frac{\iiint \in(x,y,z)\,dxdydz}{\iiint dxdydz} \tag{7}$$

For volume fraction of a fluid, three conditions are possible: (1) $\in = 0$ if cell is empty, (2) $\in = 1$ if cell is full, or (3) $0 < \in < 1$ if cell contains the interface between fluids. Then, the free surface is defined as the surface where the volume fraction is equal to 0.5, where the cell contains both air and water. Tracking of interfaces between phases is accomplished by solution of a volume fraction continuity equation for each phase:

$$\frac{\partial \in}{\partial t} + u_j \frac{\partial \in}{\partial x_i} = 0 \tag{8}$$

4.4 Boundary conditions

Since the method to predict green water effect is the dam breaking approach, there are no inlets, only opening boundary conditions. This condition allows the fluid to cross the boundary surface in either direction and it might be used where it is known that the fluid flows in both directions across the boundary. This condition is applied on the surfaces with blue arrows (see Fig. 3) with pressure equals to the atmospheric pressure.

The boundary condition for solid surfaces, such as the floor and the squared structure regions, is no-slip wall condition, which considers that the tangential velocity on the wall is null. For wall regions where the tangential velocity is not considered null, such as the side and back surfaces of the volume control, the condition of free slip wall is adopted.

The model is divided in two domains: initial water and initial air domain. Initial water domain at time $t = 0$, the volume fraction of water is 1 in all domain and the other domain is the opposite. When time $t > 0$, then water flows from the initial water domain to the initial air domain through an interface surface.

An interface cannot store momentum or energy and so, the velocity and shear stresses must be continuous through the interface. For the interface regions, General Connection interface performs the connection between two fluid domains. This can be used to connect non-correlated meshes. The GGI (General Grid Interface) algorithm is used for these cases.

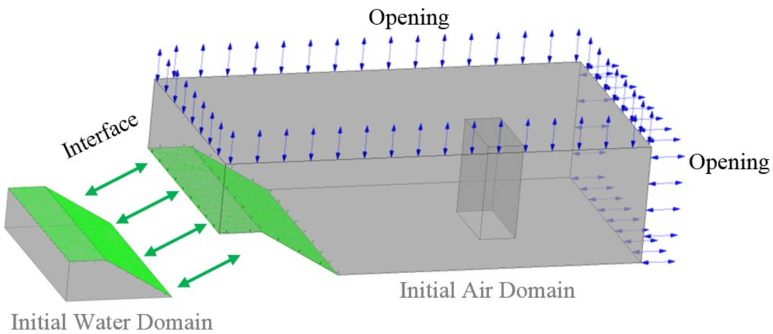

Figure 3. Boundary conditions of the model.

For the steady 2D case, the velocity at inlet can be calculated by Eq. 9 by equaling potential to kinect energy, where g is the acceleration of gravity and H is the input water height on the deck, represented by the dam breaking approach.

$$V = \sqrt{2gH} \tag{9}$$

4.5 Screening optimization method

The Screening approach is a non-iterative direct sampling method by a quasi-random number generator based on the Hammersley algorithm (ANSYS Inc., 2006). The quasi-random number generator has very low discrepancy and is used for quasi-Monte Carlo simulations. A low-discrepancy sequence is defined as a sequence of points that approximate the equidistribution in a multi-dimensional cube in an optimal way. In other words, the design space is populated almost uniformly by these sequences and, due to the inherent properties of Monte Carlo sampling, dimensionality is not a problem. The conventional Hammersley algorithm is constructed by using the radical inverse function (Latasa, 2012).

Objective functions and limits for the variables must be set in order to start the optimization process. As cited before, the optimization objective is to reduce the loads on both structures and breakwaters. The limits for the variables are defined on Table 1.

Table 1. Limits for the optimization function.

Parameter	**Limit**
Thickness (t)	10 to 20% of chord
Maximum camber (m)	10 to 30% of chord
Position of maximum camber (p)	35 to 50% of chord
Chord (c)	1.6 to 1.8 m
Angle (α)	30 to 50 degrees
Distance between foils (Δx)	1.3 to 1.6 m

5. RESULTS AND DISCUSSION

The results from the simulations for the water flow through the breakwater and the superstructure for the Configuration 1 (where the breakwater is symmetric) and Configuration 2, also for plate and NACA profiles and without breakwaters, for medium water height (10 m) can be seen on Figs. 4, 5, 6, 7 and 8. Where are showed on top view, the moment where the force on breakwaters and on the parallelepiped body is maximum, respectively. Fig. 8 shows two moments where force is maximum on the parallelepiped body, since the wave characteristics provide this type of phenomenon.

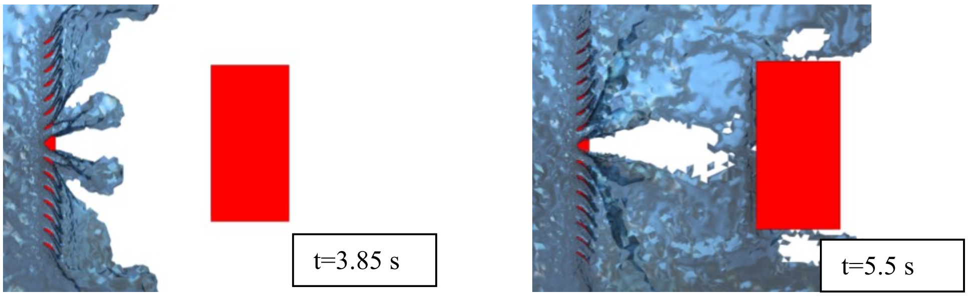

Figure 4. Water flow through the structures for Configuration 1 with NACA profile.

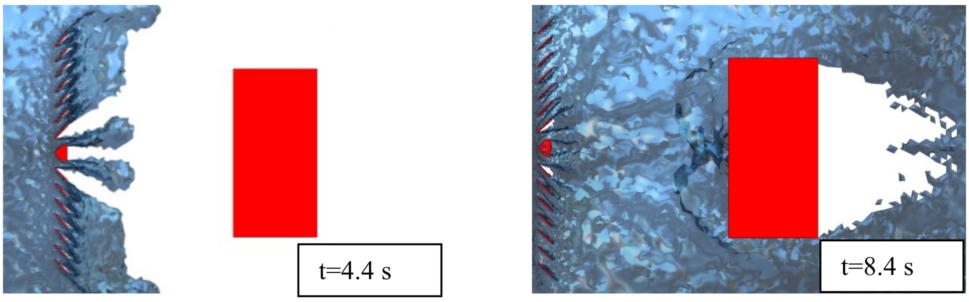

Figure 5. Water flow through the structures for Configuration 1 with plate profile.

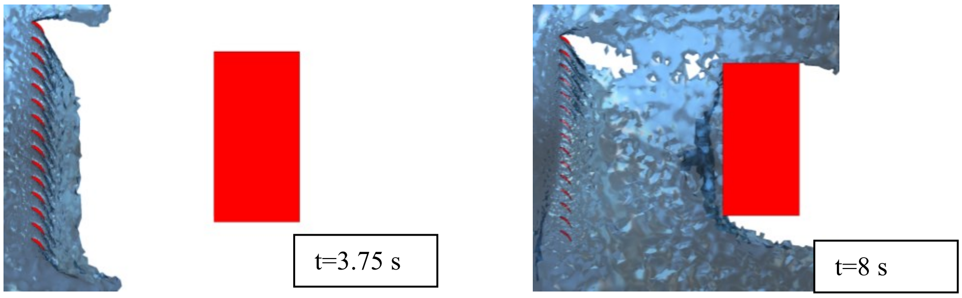

Figure 6. Water flow through the structures for Configuration 2 with NACA profile.

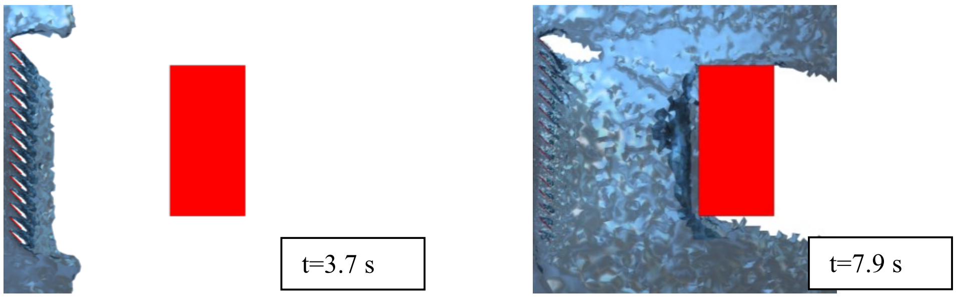

Figure 7. Water flow through the structures for Configuration 2 with plate profile.

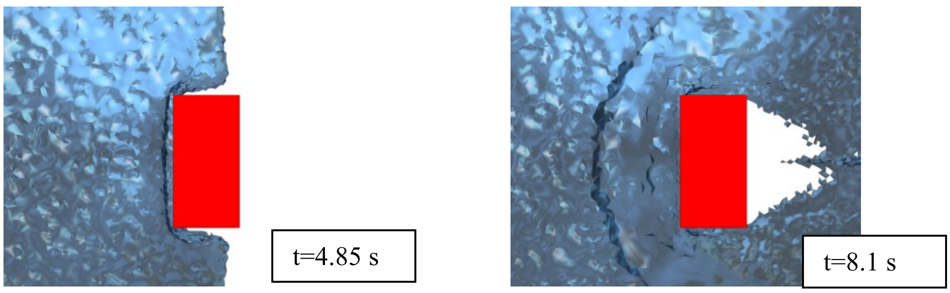

Figure 8. Water flow through the parallelepiped body on two moments of maximum force.

Graphically, these results for different water heights are shown on Fig. 9. The configuration that had the smallest value of maximum force on the breakwaters was the Configuration 2 with a NACA airfoil for all input water heights on the deck, and the one with smallest value for the parallelepiped body was the Configuration 2 with plate profile. Since the difference between the two for the force on the breakwaters is larger than the small difference on the force applied on the parallelepiped body, it was chosen for the optimization NACA airfoils.

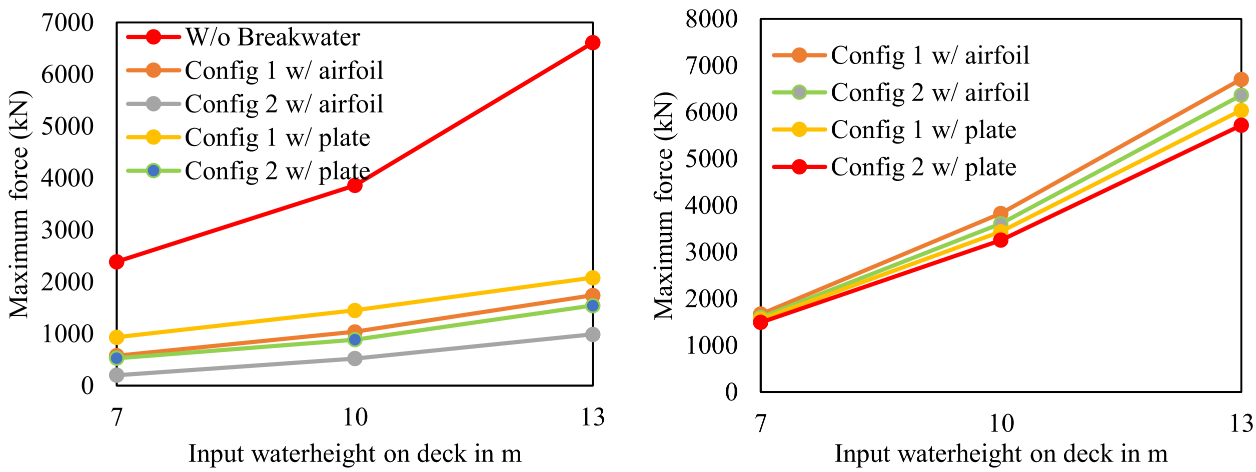

Figure 9. Loads on the parallelepiped body and on breakwater structure, respectively.

Since the largest variation on loads are done on high water heights on deck, it was used for the steady 2D simulations only the inlet velocity for 13 m. It was created 280 samples using Screening Method Optimization (Fig. 10). The optimal profile is described on Table 2.

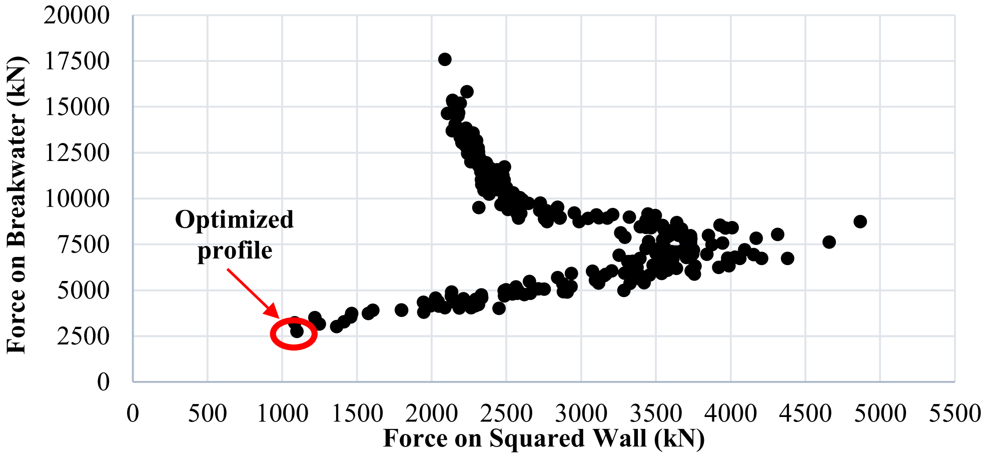

Figure 10. Loads on the squared wall and on breakwater structure for all samples.

Table 2. Characteristics of the optimized profile

Parameter	Optimized value
Thickness (t)	18% of chord
Maximum camber (m)	10% of chord
Position of maximum camber (p)	45.53% of chord
Chord (c)	1.7 m
Angle (α)	30 degrees
Distance between foils (Δx)	1.48 m

6. CONCLUSIONS

Green water effect was taken into account using a methodology that consists of simplifying the real problem, that involves complex fluid flow, body forces and interactions between the hull and water flow, into a dam breaking model, which considers a water column ahead of the structure that is released at the first time step. Other studies (Buchner, 1999; Silva and Rossi, 2012) have proven that the dam breaking model provides a good simplified model of the flow of the green water.

The dam break was modeled as a 3D unsteady free surface CFD approach, using the commercial software ANSYS CFX. The superstructure on the deck is considered as a parallelepiped body to simply the analysis. In order to reduce water impact over the superstructure, vane-type breakwaters were considered since this method is more efficient, easier to support on the deck with a lighter structure (Marin, 2003).

It was compared two configurations of the breakwaters with two different profiles and so, four different breakwaters. The profiles considered for the vanes were plate and NACA profiles due to their ease of design, since they can be designed by equations. Results for loads on the breakwaters and on the superstructures are compared with the situation were no breakwater is placed.

The configuration that has the smaller loads on the vanes and on the superstructure is selected for a goal driven optimization using a 2D steady simulation with ANSYS' screening method. Using the methodology presented, the configuration 2 with NACA profile was chosen because compared with the plate profile in the same configuration, there was a reduction of 56% of the load on the parallelepiped body and only an increase of 10% of the load on the breakwater (see Fig. 9).

An optimization on the selected breakwater configuration was performed, concerning the profile that can be modified (thickness, maximum camber, chord, angle etc). The configuration that had the minimum load on the parallelepiped structure and on the breakwater was determined (see Fig. 10 and Table 2).

An initial study for the design of breakwaters for offshore structures was realized with a consistent methodology. However, further studies must be done for real situations, considering statistical wave height appearance, interactions between water and the bodies considering 6 DOF motion, analyzing real structures for both breakwaters and superstructures in order to calculate stress due to fluid-structure interaction etc.

7. REFERENCES

ANSYS Inc., 2006. "ANSYS CFX-Solver Theory Guide Release 11.0". Canonsburg, PA.

Buchner, B., 1999. "Green water from the Side of a Weather vaning FPSO". In *Proceedings of the ASME 18th International Conference on Ocean, Offshore and Arctic Engineering*. St. Johns, Canada.

Buchner, B., 2002. *Green water on ship-type offshore structures*. Ph.D. thesis, Delft University of Technology, Delft.

Ersdal, G. and Kvitrud, A., 2000. "Green water on norwegian production ships". In *Proceddings of the Tenth International Offshore and Polar Engineering Conference*. Seattle, USA.

Hisdaris S.E. et al, 2014. "Loads for use in the design of ships and offshore structures". *Ocean Engineering*, Vol. 78.

Latasa, O.B., 2012. *Optimization of a vibration base plate by making use of finite element modeling*. Master thesis, Universidad Pública de Navarra, Navarra.

Marin, B.B. and Garcia, J.L.C., 2003. "Design aspects of green water loading on FPSOs". In *The 22nd International Conference on Offshore Mechanics & Arctic Engineering*. Cancun, Mexico.

Ryu, Y., Chang, K.A. and Mercier, R., 2007. "Application of dam-break flow to green water prediction". *Applied Ocean Research*, Vol. 29..

Shen, Y.M., 2004. "Simulation of wave propagation over a submerged bar using the VOF method with a two-equation k-ε turbulence modeling". *Ocean Engineering*, Vol. 31.

Silva, D.F.C. and Rossi R.R., 2014. "Green water loads determination for fpso exposed tobeam sea conditions". In *33rd International Conference on Ocean, Offshore and Arctic Engineering*. San Francisco, USA.

Yeganeh-Bakhtiary, A., Hajivalie, F. and Hashemi-Javan, A., 2010. "Steady streaming and flow turbulence in front of vertical breakwater with wave overtopping". *Applied Ocean Research* Vol. 32.

8. RESPONSIBILITY NOTICE