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

FREE SURFACE SIMULATION OF THE GREEN WATER EFFECT ON FPSO USING THE DAM BREAK METHOD

José Manoel Freire Guimarães
Daniel Freitas Coelho
Bruno Alberto Pignatario
Danilo de Souza Braga
Fábio Antônio do Nascimento Setúbal
Gustavo da Silva Vieira de Melo
Newton Sure Soeiro
Universidade Federal do Pará – Grupo de Vibrações e Acústica
jose.guimaraes@itec.ufpa.br
daniel.coelho@itec.ufpa.br
bruno.pignatario@itec.ufpa.br
dbraga@ufpa.br
dbraha@ufpa.br
fabioans@ufpa.br
gmelo@ufpa.br
nsoeiro@ufpa.br

Abstract. *The green water effect occurs when a ship`s bow become immersed in water and a compact mass of water wash along the deck. This is one of the main reasons of damage in ships superstructures and cargo. This paper aims to simulate and validate the green water effect and compare the results of structural loads with previous experimental data found in literature. For that, only a part of a vessel deck and a squared structure meant to be protected were built in an Ansys CFX domain, using a Computational Fluid Dynamics (CFD) approach to solve the multiphasic free surface problem with a column of water that suddenly is released over the deck. At the end, the methodology showed consistent and the results follow the same behavior of experimental data found, proving the efficiency of the computational model for the simulation of Green Water effect.*

Keywords: *Green water effect, CFD, free surface, validation, multiphasic analysis.*

1. INTRODUCTION

In heavy storms, the waves and ships motions can become so large that water flows onto the deck of a ship, causing the green water effect. The term green water is used to distinguish between small amounts of water and foam flying around and the real compact block of seawater hitting the deck (Tan, 1969). For a long time, green water has been considered an important problem for the safety and operability of natal and merchant vessels, applying destructive force to ships structures and cargo (Faulkner, 2001). In the last decade, at least twenty green water incidents have been identified on FPSOs in UK alone (Moris *et al*, 2000). Also in Norway, waves ranging from 8 m to 12.5 m were recorded and several damages to windows and walkways were taking (Ersdal, 2001).

There is several ways to solve or reduce the damage caused by the green water effect such as:

- Design the vessel and structures on the deck against the predicted green water impact load levels.
- Increase the freeboard height such that green water is prevented completely.
- Increase the freeboard height such that the green water loads are reduced to acceptable levels.
- Optimize the bow shape.
- Use protecting breakwaters in front of critical structures on the deck.
- Optimize the structures on the deck to minimize the green water impact loads.

Since its high complexity and randomness, the green water load investigation remain quite difficult and mainly addressed through experimental model and analysis as made by (Veer and Zuidscherwoude, 2012). In the other hand,

simplified analysis using numerical models were developed and proved quite useful in preliminary project modules although numerical tools usually have more difficulties to represent sea conditions.
For this reason, several authors conclude that one of the best recommendations is to use the dam break theory as a starting point to evaluate the velocity and loads from the green water phenomena. Generally, the Computational Fluid Dynamics (CFD) approach is use to solve the large number of equations with respect to the fluid domain, also a lot of research is being done to detail the simulation of green water events and there are two ways to build the vessel model. The first one is represent only a part of the vessel deck (Pham *et al*, 2005) or simulate the hull and wave interaction, this approach is more recent since more computational resources are needed to represent complex geometries and turbulence in larger models (Nielsen and Mayer, 2004).
The present work aims to simulate the dam break method using the finite volumes commercial package Ansys –CFX. The results based on the loads will be validate with experimental data found in literature.

2. MATHEMATICAL MODEL

2.1 Governing Equations

The dam-break flow is usually described by the shallow water equations, which are derived from the depth-averaged Navier-Stokes equations over the flow depth with the following assumptions:

- Incompressible Fluid,
- Hydrostatic pressure distribution
- Small Bottom slope in both x and y directions
- Uniform velocity distribution in the vertical direction

The 2D continuity and momentum equations are give as follows:

$$\frac{\partial h}{\partial t}+\frac{\partial (hu)}{\partial x}+\frac{\partial (hv)}{\partial y}=0 \qquad (1)$$

$$\frac{\partial (hu)}{\partial t}+\frac{\partial}{\partial x}\left(hu^2+\frac{g}{2}h^2\right)+\frac{\partial (huv)}{\partial y}=gh(S_{0x}-S_{fx}) \qquad (2)$$

$$\frac{\partial (hv)}{\partial t}+\frac{\partial (huv)}{\partial x}+\frac{\partial}{\partial y}\left(hv^2+\frac{g}{2}h^2\right)=gh(S_{0y}-S_{fy}) \qquad (3)$$

Where h is the water depth; g is the acceleration of gravity; u and v are the average velocity components in the $x-$ and $y-$direction that are defined as:

$$u=\frac{1}{h}\int_{z_b}^{h+z_b}\bar{u}(z)dz \qquad (4)$$
$$v=\frac{1}{h}\int_{z_b}^{h+z_b}\bar{v}(z)dz \qquad (5)$$

Where $\bar{u}$ and $\bar{v}$ are the local time-averaged velocity components. The terms S_{0x} and S_{0y} are the channel bottom slopes in the $x-$ and $y-$ direction. S_{fx} and S_{fy} are the channel friction slopes, defined as function of the Manning friction coefficient and Z_b is the bed evaluation.

Finally, the Equations (1) - (3) can be rewritten with the source terms of the shallow water equations:

$$\frac{\partial q_m}{\partial t}+\frac{\partial f_m}{\partial x}+\frac{\partial g_m}{\partial y}=S_m \qquad (6)$$

Where q_m, f_m, g_m and S_m denote the conserved physical vector, and flux vectors in the the $x-$ and $y-$ direction, and the source term respectively.

To solve the Reynolds equations, the turbulence modeling tries to represent the Reynolds stresses in terms of the time-averaged velocity components and it 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 shown in Equation (7)

$$-\rho\overline{u_i u_j}=-\frac{2}{3}k\delta_{ij}+\mu_t\left(\frac{\partial \overline{u_i}}{\partial x_j}+\frac{\partial \overline{u_j}}{\partial x_i}\right) \qquad (7)$$

where k is the kinect energy of turbulent fluctuations and μ_t is the eddy viscosity, finally it is still needed additional equations to define k and μ_t and so, obtain the Reynolds stress and this role is played by the turbulence model.

2.2 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), these works explain in details the boundary conditions applied to these equations. The two partial differential equations governing the turbulent kinetic energy k and the turbulence eddy dissipation ε then reads:

$$\rho\left[\frac{\partial k}{\partial t}+\frac{\partial(ku_j)}{\partial x_i}\right]=\frac{\partial}{\partial x_j}\left[\left(\mu+\frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right]+P_k-\mu\varepsilon+P_{kb} \tag{8}$$

$$\rho\left[\frac{\partial \varepsilon}{\partial t}+\frac{\partial(\varepsilon u_j)}{\partial x_j}\right]=\frac{\partial}{\partial x_j}\left[\left(\mu+\frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right]+\frac{\varepsilon}{k}\left(C_{\varepsilon 1}P_k-C_{\varepsilon 2}\rho\varepsilon+C_{\varepsilon 1}P_{\varepsilon b}\right) \tag{9}$$

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 $P_{\varepsilon b}$ representing 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.

$$\mu_t=\frac{C_\mu \rho k^2}{\varepsilon} \tag{10}$$

2.3 Free Surface Modeling

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:

$$\in^{cell}=\frac{\int\int\int \in(x,y,z)\,dx\,dy\,dz}{\int\int\int dx\,dy\,dz} \tag{11}$$

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:

$$\frac{\partial \in}{\partial t}+u_j\frac{\partial \in}{\partial x_i}=0 \tag{12}$$

3. THE DAM BREAK MODEL

The simplest approach to model the green water effect is consider a compact block of water with height H, width d and thickness c being suddenly released at a given time t. The gravity effect will push the water down and, consequently, forward in the structure direction. The rest of the model is composed by air. The water velocity profile will be given in terms of the water block initial parameters. In this case, the superstructure to be protected is a block with arbitrary geometric parameters. This structure could be a cargo, a navigation equipment or even a passenger deck. The Figure 1 summarizes the model and the blue arrows represent the interface between the model and the atmosphere of pressure equal to 1 atm.

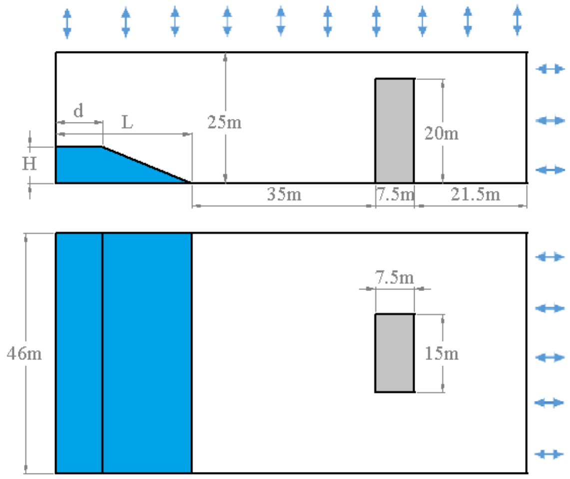


Figure 1 – Dam break model used in this work.

The model was built and simulated in ANSYS – CFX V14.5. The water domain was represented with 3 different heights of 7, 10 and 13 meters. The deck length was considered constant and with 35 meters. Other initial conditions and boundary conditions are shown below.

3.1 Computational Model and Boundary Conditions

The computational model has three different domains: air, water and solid domains. For the solid surfaces such as floor (deck) and structure, it was applied the no-slip wall condition, that mean the tangential velocity is none. In the other hand, the free-slip condition was adopted on the side surfaces of the control volume, forcing the water to remain inside the domain until it reaches the structure. The upper and back surfaces of the model were considered opening, allowing the fluid to cross the boundary surface in both.

The water and fluid domain were initially divided as shown in Figure 1 and 2. At the initial time $t = 0$, the volume fraction of water is 1 and all fluid domain and 0 in the air domain. When $t > 0$, the water flows from its domain into the air domain and still separated by a interfacial 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.

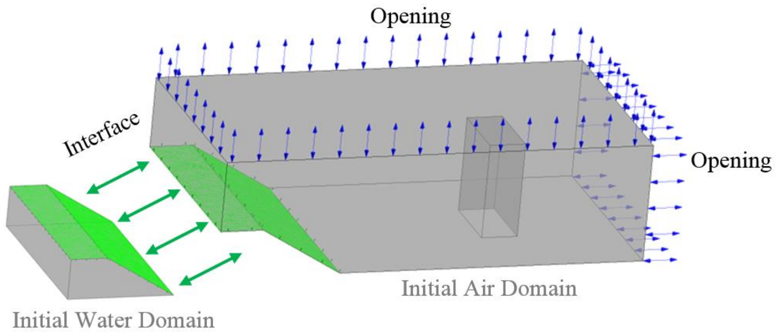


Figure 2 – Computational domain and main boundary conditions.

In addition, the dam break method does not required and inlets or outlets, only opening boundaries to assure the fluid will note reflect to the back of the structure, as the Figure 2 shows.

Another important parameter to be evaluate is the mean velocity of the fluid. It is calculate as the potential energy of the height H as:

$$V = \sqrt{2gH} \tag{13}$$

For this simulation, the tetrahedral elements were used in order to simulate the free-surface problem since it has a good adaptability to complex geometries.

Also, for a more accurate results, three different meshes were used: a coarse one with 1,420,236 elements and 258800 nodes; a medium one with 3402688 elements and 608849 nodes; and a fine one with 4460680 elements and 787879 nodes.

4. RESULTS AND VALIDATION

When the high velocity water front on the deck reaches a structure, this results in significant loading on the structure. The green water flow direction is changed after some time and there is a significant amount of water built up in front of structure which falls back on the deck after time as shown in Figure 3.

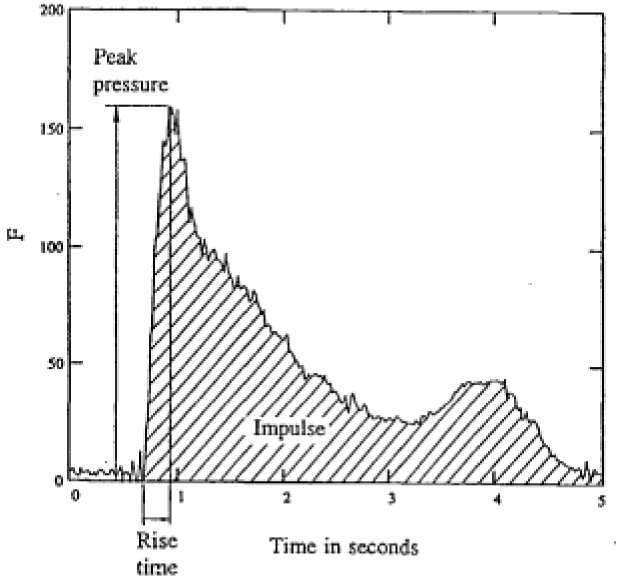


Figure 3 – Typical time trace of a green water impact pressure (kPa) (Buchner, 2002).

On the figure above, three stages can be identified:

- The stage impact: resulting in a first and highest peak of pressure.
- The quasi-static load stage: which occurs as soon as most of the kinetic energy is out of the fluid and a large amount of water has been built up in front of the structure
- The plunging water stage: when the water in front of the structure falls back onto the deck. This result in a secondary maximum peak of pressure.

As simulation result for the finest mesh, the Figure 4 shows three different times steps: the initial condition, the first impact and the plunging water stage.

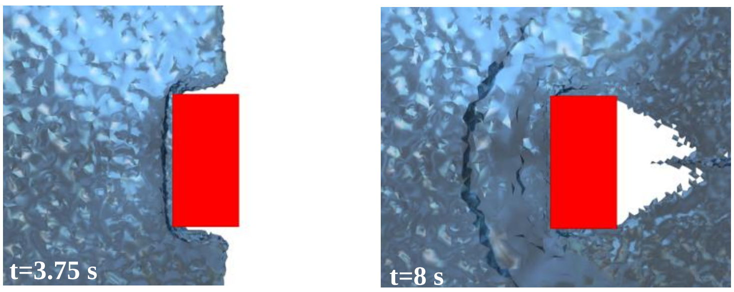
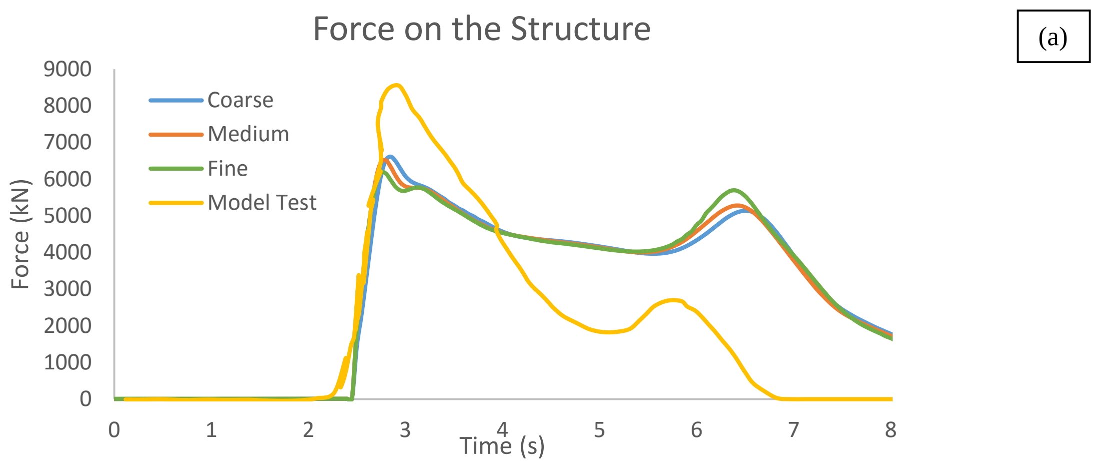


Figure 4 – Water flow onto the structure in three different time steps.

To express the results in terms of numerical load, the force and the pressure on the block was evaluated and compared to the experimental data obtained by (Buchner, 2002). The results can be observed in Figure 5 (a) and (b).



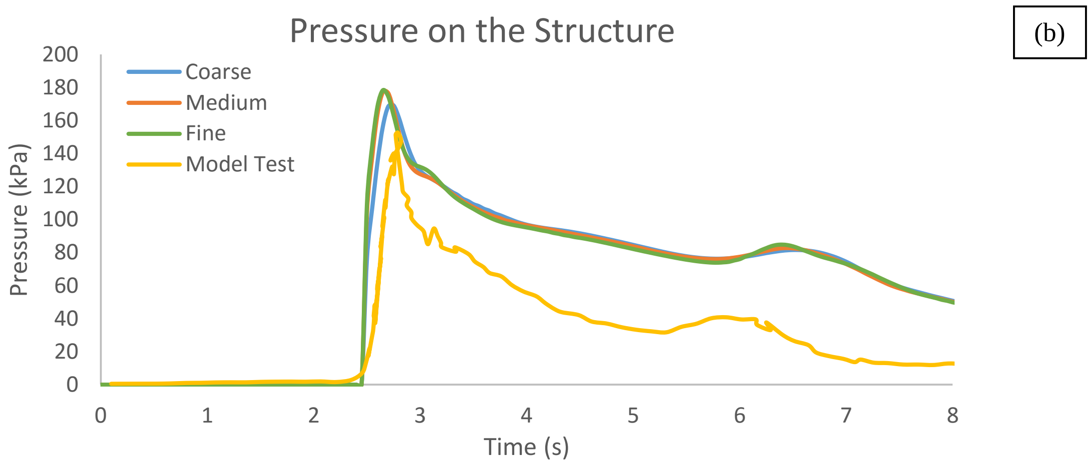


Figure 5 – Force and pressure load on the structure respectively.

The numerical results have the same behavior of the experimental data showed. When it comes to the peak value, the numerical model differ 17% in the force load and only 12% in the pressure load, which can be considered reasonable once there is a big number of simplifications adopted. The divergence of the results after the first shock peak, it may be related with the strong dissipation of the wave which propagates after the water front impacts the opposite wall, it can lead to a series of numerical noise related to the solver method.

5. CONCLUSIONS

The green water phenomena was explored at the beginning of this work and reveals itself as an important effect to be considered in offshore structures. The numerical model built presents the same generic behavior and peak conditions of the experimental data recorded by the references. However, when it comes to the after-impact behavior, the error between numerical and experimental rises. It can be consequence of simplifications adopted in the model construction and numerical noises inherent to the solution of complex problems such as the free-surface simulation.

If the maximum value is necessary to a preliminary design processes, this model presents reasonable accuracy, showing results 12 to 17% deviations in pressure and force respectively. Based on this value, it was found that the classical dam break approximation for green water analysis seems to be very useful to estimate the loads on structures, even in complex cases and geometry.

6. REFERENCES

Tan, S.G., 1969. “Observations Made on Board of Dutch Mechant Ships”. *Int. Shipbuilding Progress*, Volume 16, nº181.

Faulkner, D., 2001. “Survival Design of Cargo Hatch Structures”. *RINA Conference Design and Operation for Abnormal Conditions II*, Londres.

Morris, W.D.M., Millar, J. and Buchner, B., 2000, “Green Water Susceptibility of North Sea FPSO/FSUs,” 15th Conference on Floating Production Systems (FPS), London.

ERSDAL, G., KVITRUD, A., GREEN WATER ON NORWEGIAN PRODUCTION SHIPS. VEER, R.; ZUIDSCHERWOUDE, W., FPSO Green Wateron Deck Assessment, International Conference on Violent Flows, Nantes, France, September 25-27, 2012.

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

PHAM, X.P.; VARYANI, K.S., Evaluation of Green Water Loads on High-Speed Conteinership using CFD, Ocean Engineering 32, pp. 571-585, 2005.

NIELSEN, K.B.; MAYER, S., Numerical Prediction of Green Water Incidents, Ocean Engineering 31, pp. 363-399, 2004.

BUCHNER, B., Green Water on Ship-type Offshore Structures; PhD Thesis University of Delft, 2002.

7. RESPONSIBILITY NOTICE

The author(s) is (are) the only responsible for the printed material included in this paper.