Elangovan, Muniyandy; Sahoo, Gyanendy

Influence of Bluntness Effect on Green Water Flow in a Regular Wave

Proceedings of ninth International Conference on Hydro-Science and Engineering (ICHE 2010), IIT Madras, Chennai, India.
2 - 5 August 2010

INFLUENCE OF BLUNTNESS EFFECT ON GREEN WATER FLOW IN A REGULAR WAVE

Muniyandy ELANGOVAN.[1]and Gyanendy SAHOO[2]

Abstract: *The present study investigates the bluntness effect on Green water by introducing wave impinging on a three-dimensional simplified ship-shape structure. Numerical wave tank has been modeled with flap type wave maker and the regular wave has been generated & compared with experimental setup data. With the simulated wave, a model having a blunt angle of 45 degrees is analyzed and the flow of green water on the structure is compared with the experimental result which shows good agreement. This has been applied for the different blunt angle structure and flow has been analyzed.*

Keywords*: numerical wave tank; regular wave; CFD; model; blunt angle.*

INTRODUCTION

The interaction between extreme waves and floating structures is of primary concern in the design of offshore structures. Extreme waves not only cause stability problems but also significant damage to offshore structures due to tremendous impact forces created by wave impingement. The subsequently generated overtopping water on the deck, frequently called *Green water*, cause damage to facilities & equipment on the deck. In the past, green water problem has been investigated: phenomena of green water occurrence, influence of wave height, wave period, and current velocity (Buchner, 1995). Experimental investigations of the probability of green water occurrence as a function of Froude number & significant wave height has also been performed (Hamoudi & Varyani, 1998). The water velocities on the deck as a result of green water incidents have been studied (Shoenberg & Rainey, 2004). Although interaction of extreme waves & structures has been studied in the past, it mostly focused on finding impact loads on the structure.

Due to the development of software technology and computational power, viscous solver is used for marine applications. In this research, ANSYS-CFX has been used as a CFD solver and the authors have made an effort to study green water behavior on the deck of a ship-shape model. Effect of blunt angle of the model on the velocity field is also analyzed. The velocity field along the centerline of the structure is compared with the results of an experimental study on the kinematics of plunging breaking waves impinging and overtopping on a simplified 3D ship shaped model (Kusalika, 2009). In this research, this study is carried out by CFD & validated and additionally different blunt angle structure has been analyzed in regular wave. In general, three

1 Surveyor, Hydrodynamics and Stability Group, R&D, Indian Register of Shipping, Mumbai – 400 072, INDIA, Email: m.elangovan@irclass.org
2 Asst. Surveyor, Hydrodynamics and Stability Group, R&D, Indian Register of Shipping, Mumbai – 400 072, INDIA, Email: gyanendu.sahoo@@irclass.org

kinds of a mechanism are used in practical wave generation: plunger, flap and piston type (Wang, 1974; Tommi, 2007; Elangovan et al, 2009). For the generation of wave in the experimental setup, flap type wave maker is used. In CFD, regular wave has been simulated by flap type wave maker by our research group (Anant et al, 2009), and it was validated with theoretical and experimental data.

METHODOLOGY

Since the flap-type wave maker has been used for the study of the green water kinematics in the experimental setup (Kusalika, 2009), same is considered for the CFD analysis. Regular wave is generated in the wave maker without placing the model and wave elevations are plotted for different location for verification of regular wave. Appropriate location of the model is chosen depending on the wave height at that particular position. Model having a blunt angle of 45 degrees is placed in the required position, re-meshing is done and regular wave is generated again for further analysis. The velocity field of overtopping green water on the model is noted and compared with the experimental results. In the next step, the angle is changed and breaking wave kinematics is obtained. Comparative study of velocity field for various blunt angles is done.

MODELING OF WAVE MAKER

In the previous section, methodology was discussed to obtain and compare water velocity along the centerline of the model from experiment and CFD. Here we will discuss about the modeling of the CFD wave maker. The dimension of the tank was selected from the experimental tank size. Tank length is 25 m with a breadth of 0.9 m and height is 2 m, refer figure1. The water depth was kept at hw = 0.8 m throughout the experiments. In CFD, domain is created with the surface name called inlet, outlet, side wall, top and bottom. Here is the critical decision to select the suitable boundary condition to impose the physical condition on the domain surface. Considering our experimental tanks, inlet is flap, outlet is end wall. Side wall is kept as symmetry and the beach is placed before the end of the tank, refer figure. 2. Tank top is kept open similar to the practical which will be useful to avoid numerical instability.

The model used in the experiment was built to a scale of 1:169 of a simplified ship-shaped Floating Production Storage and Offloading (FPSO) unit and placed at 7 m from the wave maker as shown in figure 1. The angle of attack (blunt angle) of a model used in the experiment is 45 degrees. The draft of the model structure is 0.20 m and the freeboard is 0.11 m.

Flap Type Wave Maker

In this wave maker, only flap will have an angular motion with respect to the bottom of the flap. For the simulation of regular wave, constant stroke length, S_f will be provided and the flap moves sinusoidally. This kind of wave makers is used for the generation of large amplitude and long waves. Water wave is a function of angular velocity of the flap, effective water height and the stroke length. By giving suitable combination, required waves can be generated.

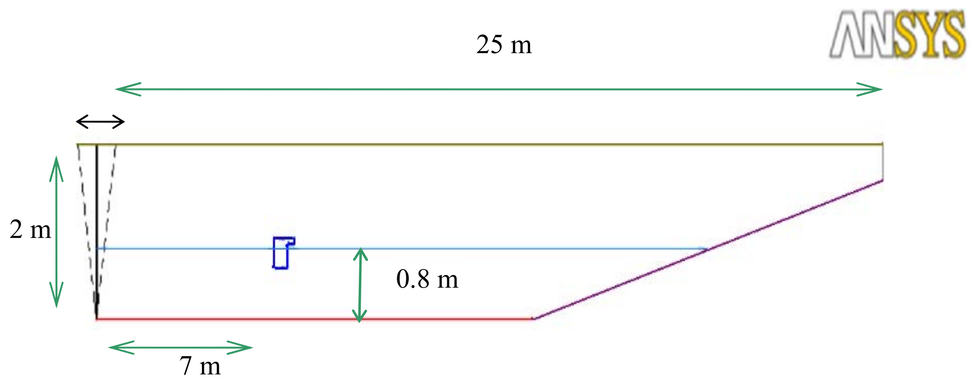
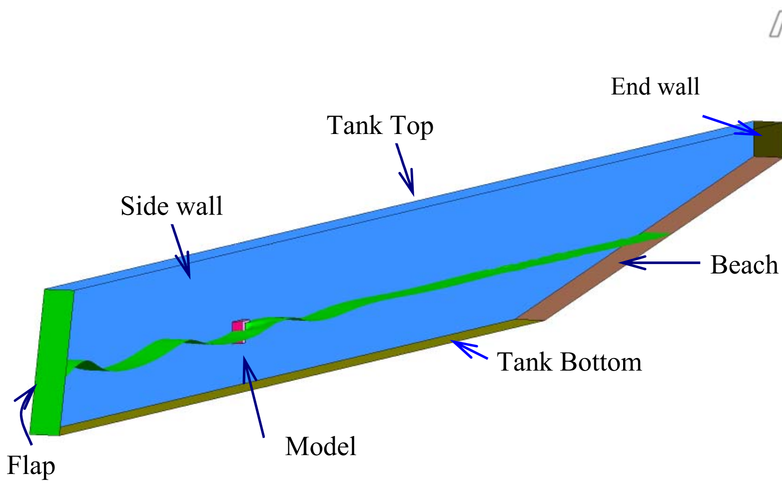


Fig.1. Flap type wave maker



0.11

Fig.2. CFD Numerical Wave maker

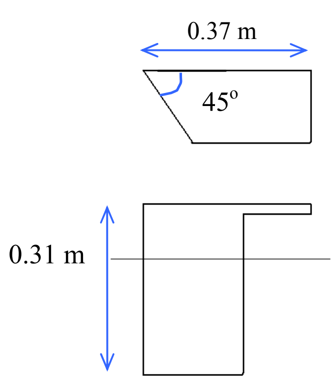


Fig.3. Model view (top and side view)

Performance and Accuracy

In numerical simulation analysis, the performance of computation and the accuracy of data can be improved by many factors like, numerical method, boundary conditions, grid size, time interval, etc. In this analysis, grid size, time interval and the turbulence are taken care for improvement which can be modified by the researcher in numerical computation analysis. Comparison for different turbulence models like k-ε, k-ω, SST has been made in the past, and it was found that SST model can capture well for near and far flow turbulence for the present application.

Wave Damping

In the wave maker tank, suitable wave damping will be provided at the end of the tank to damp the wave. Similarly in CFD, beach is provided at the end of the tank to avoid the reflection of wave or influence of wave reflection. Beach has been analyzed for different slopes like 1:3, 1:4.5 & 1:6. It was found out that 1:6 is better, but it damps slowly (Elangovan et al, 2009). Whereas for the higher slope case, damping is done quickly but small disturbance is observed. Considering all, authors have decided that lower slope will be better the wave damping.

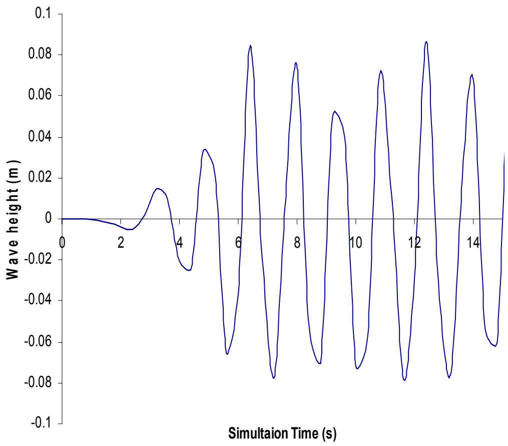
NUMERICAL SIMULATION OF GREEN WATER

The wave conditions used in the experiment and CFD wave tanks are given below in Table 1. Initially without a model, regular wave has been generated by CFD, refer figure 4. It can be seen that wave gradually reaches the required height of 0.164m at 6s. After verifying the wave details, blunt model is placed at that location and again regular waves is generated.

Wave Details	Experiment	CFD
Wave height, H(m)	0.168	0.166
Wave period, T(s)	1.36	1.5

Table.1. Wave details for both experiment and CFD

It is observed that wave front impinged the wall of the structure and overtopping of green water. When the wave front impinged on the structure, two jets were formed, and they splashed upward. The upward velocity of the water is responsible for the wave to reach the deck and the horizontal wave momentum pushed the water on the deck horizontally. The horizontal motion of the water could generate a large horizontal momentum force impact on any objects located on the deck. The green water flow on the deck of the model is shown in figure 5.



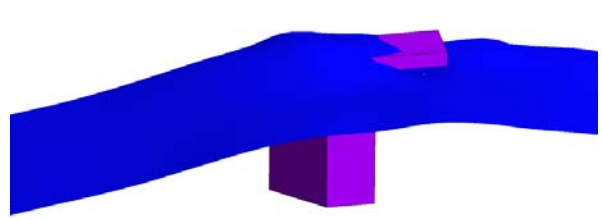
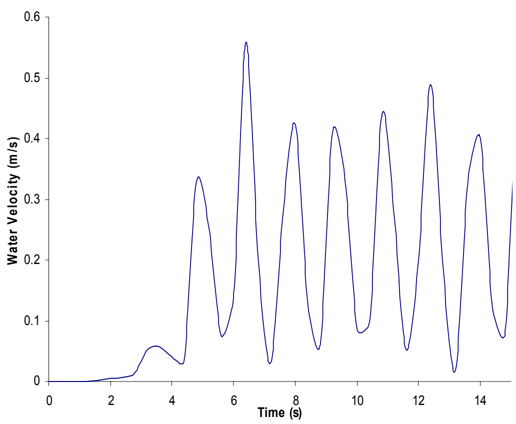


Fig.4. Wave at 7m without model **Fig.5. Green water flow on the deck of the model**

Validation with experimental data

The water velocity profile for the generated wave at 7m in the absence of the model is shown in figure 6. At t = 6 sec, the wave reaches the required height and water overtops the model. At that time, the water velocity of the wave is noted to be 0.56m/s.



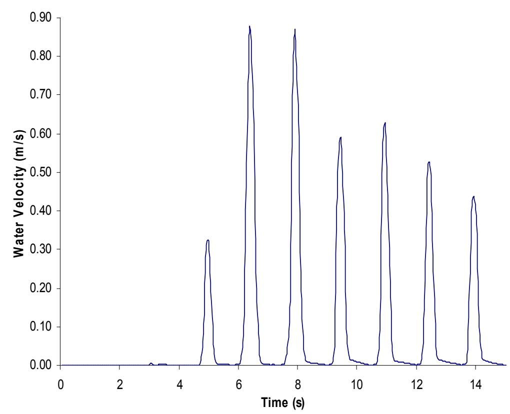


Fig.6. Water velocity profile at 7m (without model) **Fig.7. Water Velocity on the deck of the model (45 degrees)**

In the experimental wave tank, detailed velocities for the generated green water flow on top of the structure were obtained. It was observed that the maximum horizontal velocity of water on the deck of the model about 1.25-1.53 times the water velocity of the wave (in the absence of the model at that position). We also plotted velocity profile of the water on the deck which is shown in figure 7. The water velocity on the deck is observed to be 0.86 m/s i.e. 1.53 times the velocity of water in the absence of model. It is concluded that present solver can be used for the Green water flow analysis.

BLUNTNESS EFFECT

Until now we have seen the generation of regular wave in the numerical wave tank and simulation of green water flow on the model with blunt angle of 45 degrees. To study the influence of bluntness effect on green water flow, ship-shaped models are made with different blunt angle i.e., 10, 15, 30, and 90 degrees. Four CFD domain model is modeled for each blunt angle and analysis is carried out with same wave details.

1) Blunt angle - 10 degree:

The angle between centerline of the model and adjacent side is kept at 10 degrees. Velocity field on the deck is obtained for this case. It was found out that the velocity along the deck at 6 sec was 0.452 m/s. Previously it has been discussed that the velocity field at t = 6 sec is only considered because incident wave is not affected by reflected wave at that time. In this case, the velocity field in subsequent 3 cases is nearly same to that found for t = 6 sec. Since the angle of attack of the model is less, reflection of wave is not so prominent. Hence, velocities for consecutive 4 wave fronts (from 6 sec to 12 sec) are almost equal.

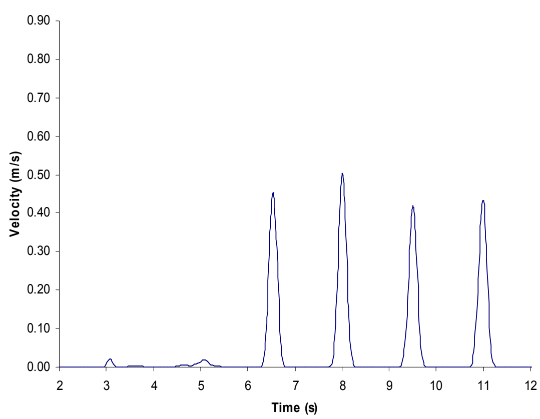


Fig.8. Velocity field for θ = 10 degree

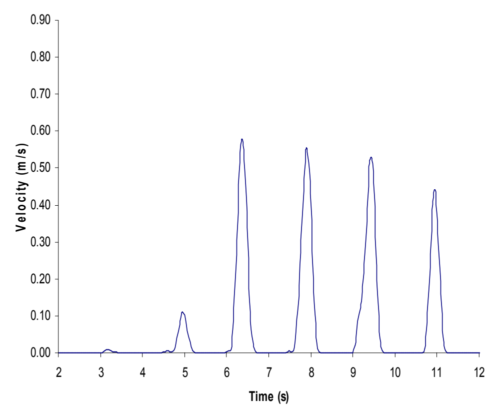


Fig.9. Velocity field for θ = 15 degree

2) Blunt angle – 15 degree:

In the second case, angle of the model is changed to 15 degrees. Velocity of the overtopping water on the deck is found to be 0.578 m/s. But the velocity field for the next 3 wave fronts decreases gradually. This phenomenon can be explained by the reflection of waves from the model. The velocities of green water flow on the deck for the blunt angle 15 degree are shown in figure 9.

3) Blunt angle – 30 degree:

In this case, blunt angle (θ) of the model is made 30 degrees. But the velocity of flow along centerline of the model at 6 sec is found to be more than that of 10 & 15 degrees. However, when the next wave front flows on the deck, velocity is more than that of the first wave may be due to the accumulation of water near the model wall. Here velocity field varies largely from 6 sec to 12 sec, unlike the previous cases where velocities were nearly same.

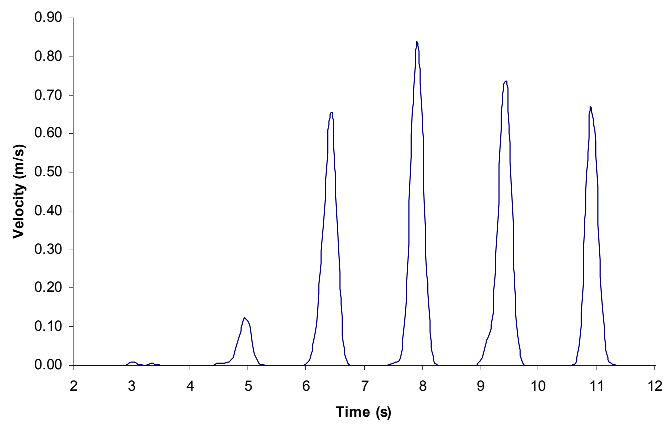


Fig.10. Velocity field for θ = 30 degree

4) Blunt angle – 45 degree:

The blunt angle of the model is changed to 45 degrees in this case. This model has been used previously for the comparison with the experimental wave tank results. Velocity field obtained for this model was found to be greater than previous ones. Velocity for the flow on the deck at t = 6 sec is 0.86 m/s. But after the first two wave fronts pass over the deck, velocity profile for subsequent waves decreases drastically, which is shown in figure.7.

5) Blunt angle – 90 degree

In this case, the angle of attack of the model is kept as 90 degrees, which are rectangular in shape as seen from top view shown in figure 11. Velocity profile obtained for this model shows that the velocity for the first overtopping wave is maximum and then decreases gradually for the remaining waves. The reason behind this occurrence is the front shape of the model which reflects the waves. Hence the height of the incident waves decreases for subsequent waves due to the damping effect by reflected waves. The velocity profile for this model is shown in figure 12.

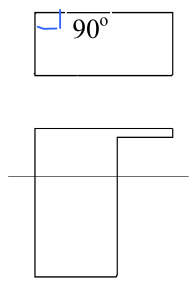


Fig.11. Model with blunt angle = 90 degree

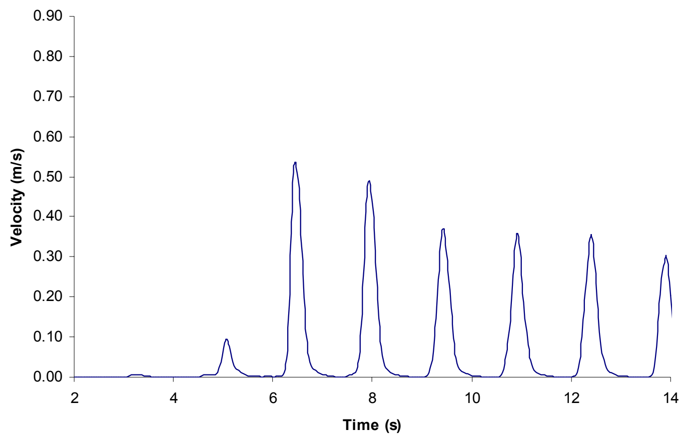


Fig.12. Velocity field for θ = 90 degree

CONCLUSIONS

Regular waves are generated in flap-type wave maker in the CFD wave tank. Placing the ship shaped model in the wave tank, velocity field of Green water flow on deck along the centerline of the model is determined. The ratio of water velocity with model and without model is obtained from CFD was found to lie in the desired range as in an experimental tank i.e. 1.25-1.53. Later, velocity profile for models with different blunt angles is obtained to study the effect of bluntness on green water flow. It is concluded Green water velocity on the deck is increasing as the blunt angle reaches till 45 degree and it comes down. Blunt angle 45 degrees has the higher velocity profile when compare to all other blunt angle. Blunt angle of 90 degrees, velocity decreases gradually after the first peak due to damping of wave by the reflected wave from the model wall. Present analysis, body is in fixed location and restricted with body motion. In future, bluntness effect by CFD will be carried out with 6 DOF and forward speed.

ACKNOWLEDGEMENTS

Authors would like to thank Indian Register of Shipping or the continual encouragement towards the research and permission for the publication of paper.

REFERENCES

Anant Lal and M. Elangovan, "CFD Simulation and Validation of Flap Type wave maker", *International Conference on Computational Fluid Dynamics* (WASET 2008), Thailand, 17-19 December, 2008.

B Buchner (1995), "The impact of green water on FPSO design", *Offshore Technology Conference 95,* OTC, Vol, 7698, pp. 45-57.

B Hamoudi and KS Varyani (1998), "Significant Load and green water on deck of offshore vessels", *Ocean Engineering,* 25, pp 715-731.

D.C. Wan and G.X. Wu, "The numerical simulation of the green water effect", Shanghai

University, Shanghai, China

Elangovan Muniyandy and Anant Lal (2009), "Study and Analysis of Sea wave Generation by wave maker", *International Conference in Ocean Engineering (ICOE),* Chennai, India, 1-5 February.

Kristian Bendix Nielsen and Stefan Mayer (2004), "Numerical prediction of green water incidents", *Ocean Engineering,* pp 363-399.

Kusalika Ariyarathne, Kuang-An Chang and Richard Mercier (2009), "Measurement of Green Water on a 3D Structure", *International Conference on Ocean, Offshore and Arctic Engineering,* Honolulu, Hawaii, USA, May 31-June 5.

M. Greco, O.M. Faltinsen, M. Landrini, "Green water loading on a deck structure"

T Schoenberg and Rainey (2004), "A hydrodynamic model of Green water incidents", *Applied Ocean Research,* 24, pp 299-307.

Tommi Mikkola (2007), "Simulation of Plunger Type Wave makers", *Journal of Structural Mechanics,* Vol 40, No.4, pp 19-39.

X.P. Pham and K.S. Varyani (2005), "Evaluation of green water loads on high-speed containership using CFD", *Ocean Engineering*, pp 571-585.

Yonguk Ryu and Kuang-An Chang (2005), "Breaking Wave Impinging and Green water on a Two-Dimensional Offshore Structure", *International Offshore and Polar Engineering Conference,* Seoul, Korea, June 19-24.

Yonguk Ryu, Kuang-An Chang and Richard Mercier (2007), "Runup and green water velocities due to breaking wave impinging and overtopping".