Elangovan, M.; Kumar, J. Ram

Becnh Mark Study on The Use of Computational Fluid Dynamics for Marine Application

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

BECNH MARK STUDY ON THE USE OF COMPUTATIONAL FLUID DYNAMICS FOR MARINE APPLICATION

M. ELANGOVAN.[1],and J. Ram Kumar[2]

Abstract: Presently, many commercial computational fluid dynamics' codes are available in the market for the analysis of an engineering problem especially in marine application. In the present research, using the CFD software, the benchmark problems have been analyzed for the slamming load, Green water load, wave run-up and tank sloshing load estimation. Each problem has been modeled by CFD and compared with experimental & computational results.

***Keywords** wedge impact; dam break; CFD; model; sloshing.*

INTRODUCTION

Nowadays, new ship / offshore structures are analyzed by the experiment or numerical method before starting construction to validate/confirm the performance. The numerical methods widely used are based on potential theory, which assumes the fluid is invisicid, incompressible and irrotational. These methods are considered as fast and robust tools in the design stage as they allow a large number of parameters to be analyzed for the purpose of optimizations. The motion of simple bodies in waves can be computed using these methods with reasonable accuracy. However, potential theory based software are yet to improve, where viscous effects or breaking waves play an important role such as slamming, sloshing and green water.

Though few commercial viscous solvers are available in the market for the analysis of marine related problems and it is very important to validate with bench mark problem before applying to the practical problems. In our present research, the Finite Volume Method commercial code, ANSYS-CFX 11.0 has been used to carry out the above said bench mark problems. This numerical approach involves discretizing the spatial domain into finite control volumes using a mesh. The governing equations of mass and momentum are integrated over the control volume, such that the relevant quantities are conserved. The Volume of Fluid (VOF) method has been used to capture the free surface which was originally proposed by Hirt and Nichols (Hirt & Nichols, 1981).

Marine structure failures are analyzed by many researchers around the world by CFD. Von Karman (Van, 1929) was the first researcher to study water impact (slamming) theoretically. He idealized the impact as 2-d wedge entry problem on the calm water surface to estimate the water

1 Surveyor, Hydrodynamics and Stability Group, Research and Development, Indian Register of Shipping, Mumbai – 400 072. INDIA, Email: m.elangovan@irclass.org
2 Asst. Surveyor, Hydrodynamics and Stability Group, Research and Development, Indian Register of Shipping, Mumbai – 400 072. INDIA, Email: j.kumar@irclass.org

impact load on a seaplane during landing. Since the Impact is so rapid, Von Karman assumed very small water surface elevation during impact and negligible gravity effects. Since Von Karman impact model neglects the water surface elevation, the added mass and impact load are underestimated, particularly for small dead rise angles. Wagner modified the Von Karman solution by taking into account the effect of water splash on the body (Wagner, 1936: Korobkin and Pukhnachov, 1988). Zhao et al (1993, 1996) introduced a complement to Wagner's study, with linear approximation of the free-surface boundary conditions for the two- dimensional problem. The variation of the body velocity due to impact is not considered by aforementioned authors. Mei et al (1999) considered the effect of the variation of the body velocity is taken into account in determining instantaneous pressure distribution.

For studying the Wave run up effect dam break problem can be taken as a bench mark study. Stocker (1957) was the first to do works on a dam break problem. The problem can be simply defined as a closed space with full of water. Study of flow is now done by removing the barrier. In marine transportation, oil carriers are used to carry large amount oil from between many countries. In carriers, partially filled tanks can be violent if the period of exciting motion of the tank is close to the natural period of fluid inside the tank. Although it is undesired, it is unavoidable to have partially filled tanks in ships. Therefore it becomes important to estimate dynamic loads in partially filled tanks. The experiments were conducted by Hiantsu-et-al (2001).

In ocean going vessel, extreme waves cause significant damage to offshore structures due to tremendous impact forces created by wave impingement and the subsequently generated overtopping water on the deck, frequently called Green water (Kristian 2003 & 2004,), cause damage to facilities & equipment on the deck. Here authors have made effort study the above problems by CFD software. Each problem has been modeled in CFD and analyzed each problem and compared with sufficient experimental results and or computational published results.

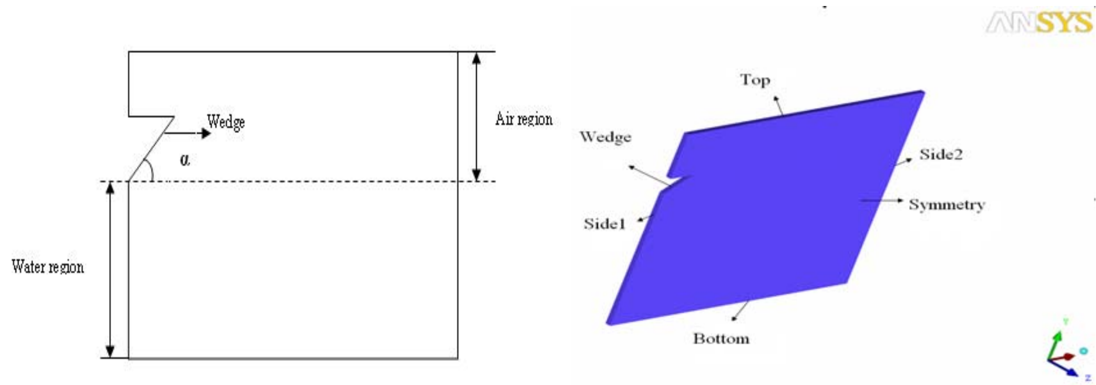
SLAMMING LOAD ANALYSIS

To estimate the slamming load, researchers are starting with 2Dsection with wedge shaped structure. Presently, the impact load analysis is done in two cases where wedge and cylinder are used in an analysis. In the wedge impact problem, the variation of wedge velocity due to impact load calculated analytically and the loads are estimated. In the case of a cylinder impact, cylinder is moved with uniform velocity and the estimated loads are compared with experimental results.

Case 1: Wedge Impact

Modeling

Due to the symmetrical nature of the wedge, only one half of the wedge is considered for the analysis. Wedge with dead rise angle (α) 30 degrees, mass 154 kg and drop height of 1. 3m is considered for analysis. The length of the tank is 6 m and the water height is taken to be 3 m. To reduce the computational effort, 2D analysis is done for wedge with dimensions 0.6 m x0.1m. The schematic diagram of wedge with water surface is shown in figure 1(a) . Three dimensional view is shown in figure 1.(b) with higher density of mesh.



(a) (b)

Fig.1. Wedge Impact Experimental and CFD Model

Analysis

Calculations for Reduced velocity:

The instantaneous velocity of the wedge during its impact period is given (EI-M, Yettou, 2006) by

$$V(\text{t}) = \frac{V_0}{1 + \frac{M_a}{M}}$$

Where, M is the mass of the wedge, V_0 is the initial velocity (at t = 0), M_a is the added mass.

$$\text{M}_a = \text{C}_a \, \rho \, (Y(\text{t}))^2$$

Where $\text{C}_a = \frac{\delta\pi}{2}(1 - \frac{\alpha}{2\pi})^2$

Y(t) represents the horizontal coordinate of the intersection between the body and the free surface, α is the dead rise angle and δ is a correction factor taking into account the three dimensional effects associated with the no infinitely long bodies used in the experiments, it varies from 0 .5 to 1(Zha0 et al 1996).

Y(t) is solved using

$$\frac{\rho C_a}{3} Y(\text{t})^3 + M \, Y(\text{t}) - \frac{\gamma M V_0}{\tan(\alpha)} \text{t} = 0$$

γ is a dimensionless parameter that depends solely upon the dead rise angle of the wedge; it measures the waters splash onto the wedge. It can be easily provided from the equation given by Mei (Mei et al 1999). The above given cubic equation is solved to get Y(t) with respect to time. Substituting the Y(t) in the velocity equation, V(t) is calculated and this will be transformed into the displacement which will be given as input to the CFD. The displacement is plotted and shown in figure 2. The instantaneous force on a wedge is calculated from CFX and compared with the experimental result which is shown in figure 3. Though the numerical are matching well with experiment, numerical results predict little more. This is to be analyzed well in future.

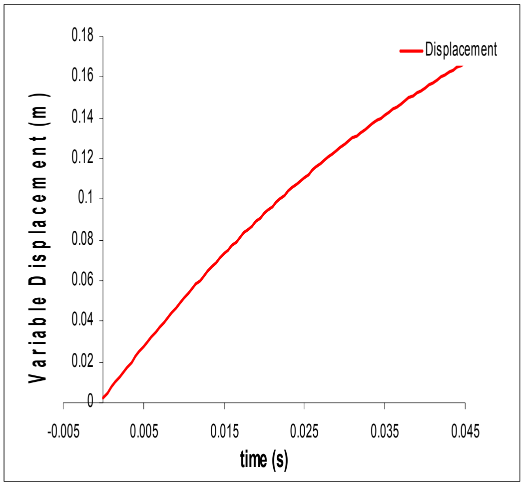


Fig.2. Variable wedge displacement

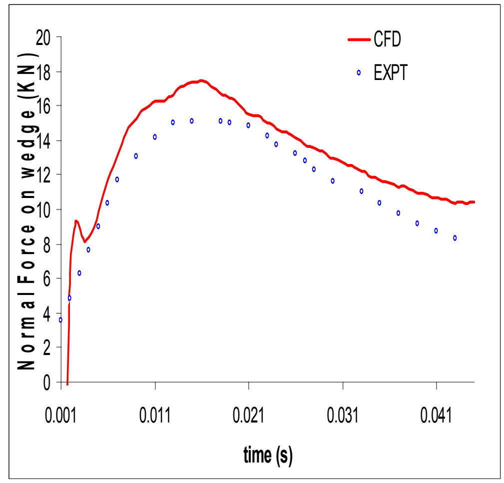


Fig.3. Comparison of Force on wedge

Case 2: Cylinder Impact

Modeling

In this analysis, cylinder is dropped from center height and it moves with the same velocity. The dimension of a tank is 6 m x 0.4 m x 5 m. The cylinder has a diameter d=0.35 m and length l = 0.40 (m). Two drop height is selected based on that velocity is calculated when it reaches the water surface. The selected impact velocities are V= 0.9 m/s and V= 1.35 m/s. The schematic diagram and the CFD modeling are shown in figure 4.

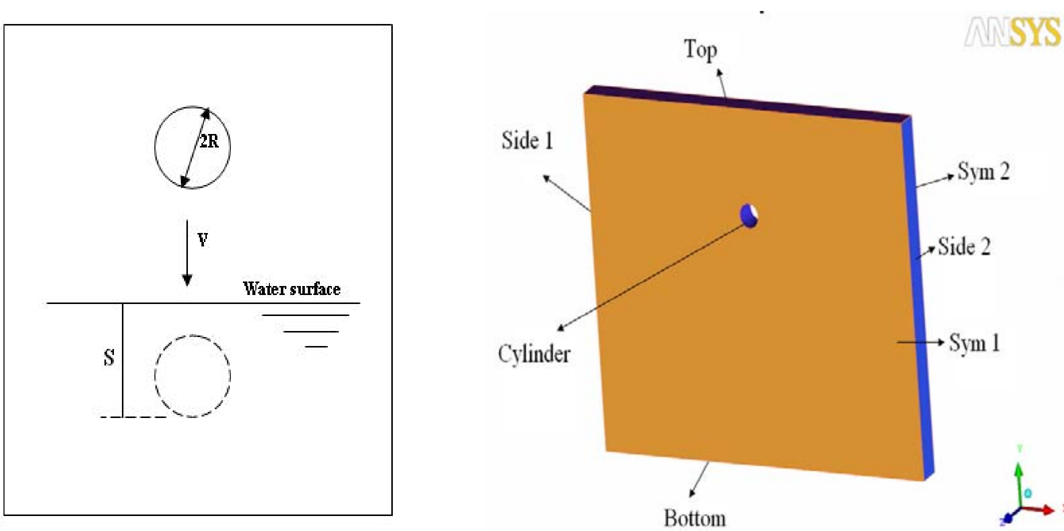


Fig.4. Cylinder Impact Experimental and CFD Model

Analysis

The load on the cylinder due to impact for both velocities is computed through CFX and compared with the experimental results (Faltinsen et al 1977, Kristian, 2003). Numerical results shows that CFD can predict well when compare to the experimental values. It is understood that CFD can be used for slamming load analysis.

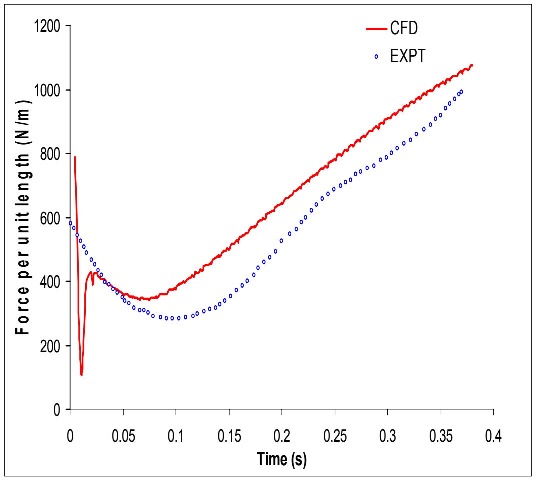


Fig.5. Force on cylinder for v=0.9 m/sec

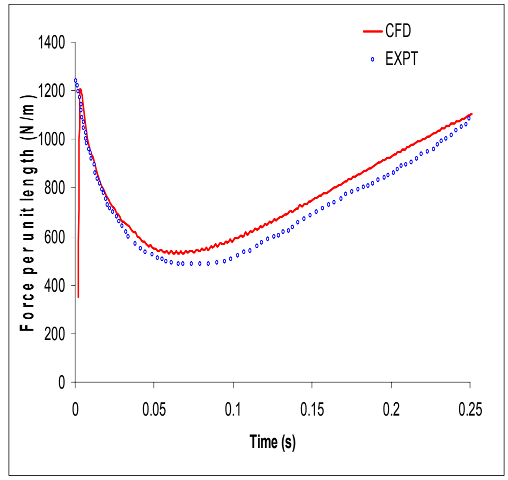


Fig.6. Force on cylinder for v=1.35 m/sec

WAVE RUN-UP ANALYSIS

Modeling

The problem can be simply defined as a closed space with full of water. Study of flow is now done by removing the barrier i.e., flap. The initial water height is equal to 0.6 m and tank length is 3.2 m with the width equal to 1m.

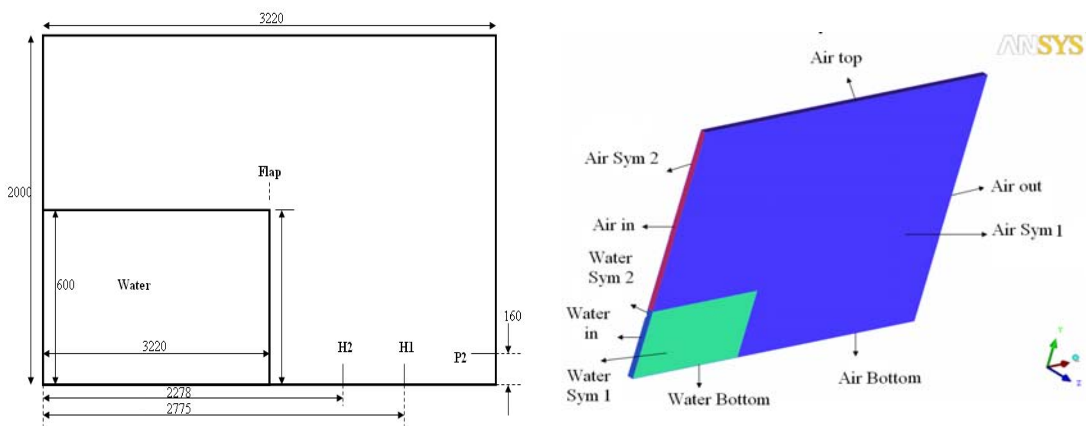


Fig.7 Wave run -up Experimental and CFD Model

Analysis

The test was initiated when the flap was lifted, and the water could freely flow into the void, and impact on the wall on the right side of the tank. The water height is measured at location H1 where the probe is located. The comparison of free surface at probe 1 of both experimental and CFD is shown in figure 8.The figure shows a good agreement between experimental and computational results at the initial stage of the test. However, when the water height increases after the water return from impact with the wall, the computed water height is significantly higher than the experimental result. After a short period the water height decreases rapidly and drops to a level below the experimental results(Stoker,J.J. 1957, Kristian, 2003)

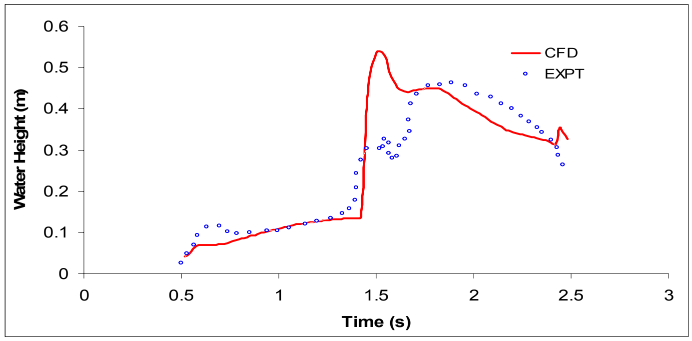
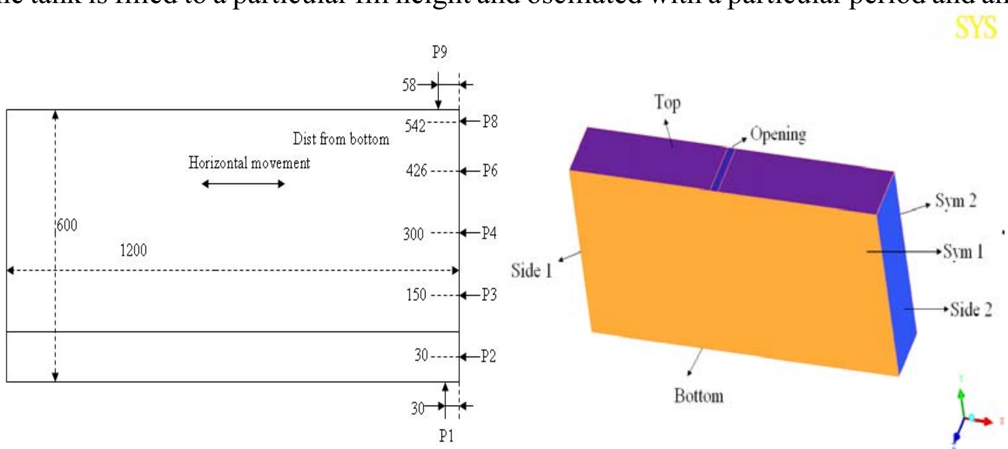


Fig.8. Water run up at probe H1

SLOSHING - TANK

Modeling

A rectangular tank with the dimensions of 1.2x0.6x0.2 m is modeled. Pressure gauges are placed at different heights as shown in figure 9 (a). The tank is given a horizontal sinusoidal motion. The tank is filled to a particular fill height and oscillated with a particular period and amplitude.



(a) (b)

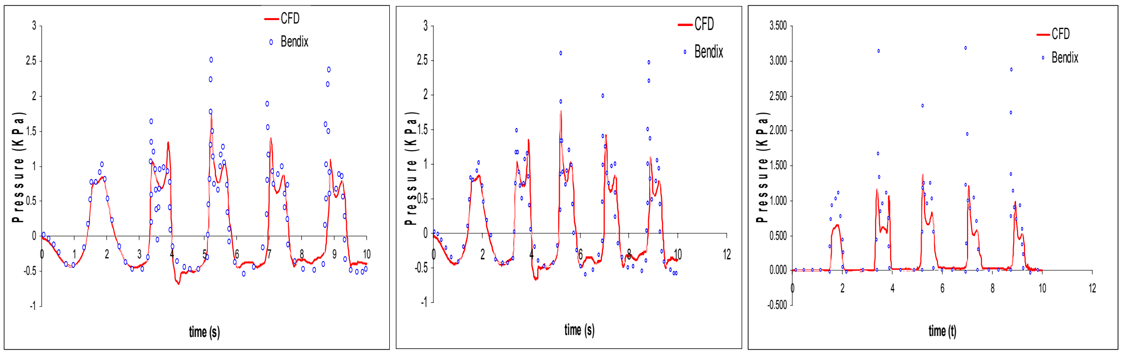
Fig.9. Sloshing Tank Experimental and CFD Model

Analysis

The tank is filled up to 20% fill level. And the tank is oscillated with a time period of 1.74 s. The amplitude of tank motion is given to be 0.06 m. In CFD, horizontal sinusoidal motion can not be given. Therefore, the motion is transformed as a horizontal acceleration,

$$\ddot{x} = -A\omega^2 \sin(\omega t)$$

The simulation is done for 10 s and the pressures at points P1 and P2 and P3 are compared with the experimental results. For pressures at a point P1 & P2 the static pressures are deducted from the CFD pressures and the results are matching well with the computational results (Kristian 2003).



(a) **(b)** **(c)**

Fig.12. Comparison of pressure at point 1,2 &3

GREEN WATER LOAD ANALYSIS

Modeling

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 2 m, refer figure1. The water depth was kept at hw = 0.8 m throughout the experiments. The model structure was located at 7 m from the flap of the wave maker. The angle of attack (blunt angle) of a model used in the experiment is 45 degrees. The model was set inside the wave tank as shown in figure 13. Therefore, the problem is three-dimensional. The draft of the model structure is 0.20 m and the freeboard is 0.11 m.

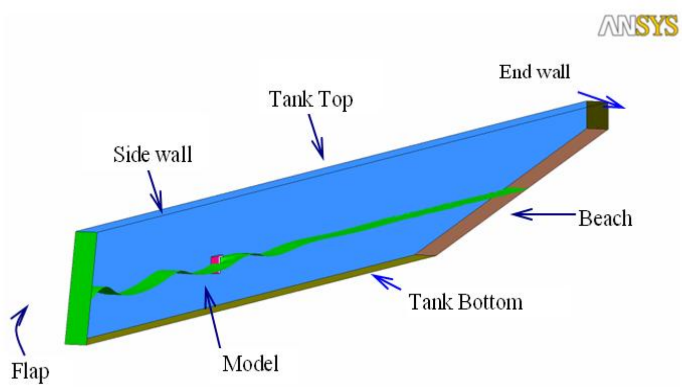


Fig.13. CFD Numerical Wave maker

Analysis

The wave conditions used in the experimental setup in the wave tank is of a wave height of 0.168 (m) and wave period of 0.136(s), in CFD waves are generated with a wave height of 0.166 (m) and wave period of 0.15 (s). 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.

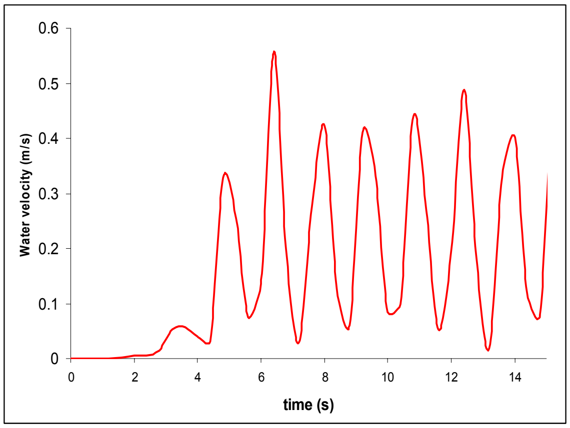


Fig.14. Water velocity profile at 7m (without model)

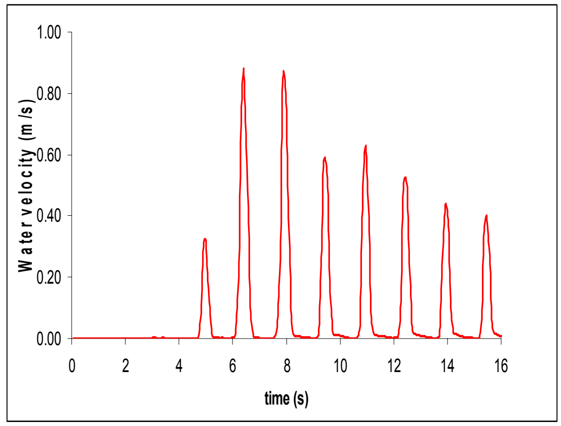


Fig.15. Water Velocity on the deck of the model (45 degrees)

In the experimental wave tank, detailed velocities for the generated green water flow on top of the structure were successfully obtained. The maximum horizontal velocity of water on the deck of the model was found to be 1.25-1.53 times the water velocity of the wave (in the absence of the model at that position). In the CFD wave tank, the velocity profile of the water on the deck of the model is obtained and compared with the experimental results (kusalika, 2009). The green water velocity on the top of the model is shown in figure 15. At t = 6 sec, 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 the model.

The velocity field of green water flow on the deck is obtained only for the first wave front that passes over the model. This result is used for comparison and validation of green water flow with experimental data. Subsequent waves are affected by the reflection of water from the model wall and wave height changes. Hence, results obtained at t = 6 sec, is only taken into account for comparison.

CONCLUSIONS

CFD has been applied for the analysis of slamming, green water, wave run-up and sloshing problems. Each case, modeling and meshing is done by ICEM CFD software. Slamming is validated with cylinder impact and wedge impact analysis. Wave run-up study is made and compared with experimental data. Sloshing is validated with experimental data and it can be applied for real practical problems. Similarly for the Green water load analysis, Bench mark problem has been analyzed and compared with experimental data. Author would like to conclude that CFD software can be used for the analysis of marine problems and this can be applied for the proto type to get real sea state behavior of the vessel and the load particulars.

ACKNOWLEDGEMENTS

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

REFERENCES

El-M. Yettou, A. Desrochers, Y. Champoux , " A new analytical model for pressure estimation of symmetrical water impact of a rigid wedge at variable velocities" *„Journal of Fluids and Structures*. October (2006).

Faltinsen, o., Kjaerland, O., Nottveit, A. and Vinje, T. (1977). Water impact loads and dynamic response of horizontal cylinders in offshore structures. In *Offshore Technology Conference*.

Hiantsu, M., editor (2001). Proceedings of SRI-TUHH mini-Workshop on Numerical Simulation of Two-Phase Flows. *National Maritime Research Institute*, Tokyo, Japan.

Hirt, C.W., and Nichols, B.D., "Volume of fluid method for the dynamics of free boundaries", *Journal of computational physics*, 39, 201, 1981

Korobin, A.A., Pukhnachov, V.V., 1988, 'Initial stage of water impact', *Annual review of fluid mechanics*, 20, pp 185-195.

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

Kristian Bendix Nielsen(2003),"Numerical prediction of Green water loads on ships", *PhD Thesis*, Technical University of Denmark.

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.

Mei.X., Lui,Y., Yue, D.K.P., 1999, On the water impact of general two-dimensional sections. *Applied Ocean Research* 21,1-15.

Stoker, J.J. (1957). Water Waves. In Pure and Applied Mathematics, Edited by R. Courant, L. Bers and J.J. Stocker. *Interscience*, New York.

Van Korman, T., 1929, The impact on seaplane floats during landing, *Technical notes*, National advisory committee for Aeronautics.

Wagner, H., 1936, Phenomena associated with impacts and sliding on liquid surfaces, NACA library, Langley aeronautical laboratory.

Zhao, R., Faltinsen , O.M., 1993. Water entry of two-dimensional bodies. *Journal of Fluid Mechanics* 246, 593-612.

Zhao, R., Faltinsen, O.M., Aarsnes, J., 1996, 'water entry of arbitrary two dimensional sections with and without flow separation', *In. Proceedings of the 21st symposium on Naval hydrodynamics,* Trondheim, Norway, National Academy Press, Washington, DC, USA.