# CFD ANALYSIS OF NON-LINEAR SLOSHING IN A CYLINDRICAL TANK

**Shruti Baghel[1], Subhash Shinde[2]**

[1]*M.Tech Student, Department of Aerospace Engineering, UPES, Dehradun, India*
[2]*Senior Engineer, Fluidyn Consultancy Pvt. Ltd., Bangalore, Karnataka, India*
***shrutibaghel92@gmail.com***

## Abstract

*Present study involves CFD analysis of non-linear sloshing inside a partially filled cylindrical tank under harmonic excitation with two different frequencies using commercial software suite Fluidyn MP.Water-air interface is captured through volume of fluid (VOF) multiphase model using high resolution interface capturing scheme (HRIC). Flow is considered as laminar and full gravity model is implemented. Obtained numerical results in terms of wave height or sloshing height are compared with experimental values given by Yung - Hsiang Chen et.al. [1]. Obtained numerical results for two different excitation frequencies are observed to be showing good qualitative and quantitative agreement with experimental results.*

***Keywords: -*** *Fluidyn MP, CFD, VOF, HRIC*

---------------------------------------------------------------------***---------------------------------------------------------------------

## 1. INTRODUCTION

Liquid sloshing is nothing but the wave motion of liquid subjected to some kind of oscillation. Sloshing phenomenon is complex movement of fluid and from engineering design and liquid transport safety point of view it has a great importance. Liquid can be set to the violent oscillation when it is subjected to external excitation having frequency close to natural frequency of sloshing that can result into permanent damage of the structure. Hence it is necessary to study the sloshing behavior of liquid under different excitation scenarios and violent sloshing behavior should be avoided.

Computational fluid dynamics (CFD) is found to be sophisticated technology for carrying out such type of analysis, saving overall cost and time and hence augmenting the design procedure. The capability of getting the solution in limited time with the use of high performance computers brought the importance to computational fluid dynamics (CFD). CFD involves the solution of basic equations of motion for a given flow field resulting into the study of basic to complex nature of flow and its properties.

Many researchers have both experimentally and numerically studied the sloshing behavior inside partially filled tanks. P. Pal [2] experimentally studied the liquid sloshing behavior inside a partially filled rectangular container and compared same with numerical results. P. Pal [2] concluded that slosh characteristics are strongly influenced by the tank geometry, depth of liquid, the amplitude and frequency of external excitation. It is observed that steepness has important effect over linearity of the sloshing and on the wave amplitude [3].

Effect of baffle positioning inside the tank over sloshing behavior is studied by Chung-Yueh Wang et. al. [4] who has carried out the numerical simulation of liquid sloshing behavior inside rectangular tank with and without baffles. From their study it is observed that the maximum free surface displacement decreases as the height of location of the baffles increases. Chung-Yueh Wang et. al. [4] also mentioned that, for single vertical baffle or two vertical baffles placed in a rectangular tank, the optimal position of placing of baffle or baffles for the reduction of the sloshing motion of fluid is located at the center of the tank. O. R. Jaiswal et. al. [5] mentioned the importance of numerical techniques in order to find out the sloshing frequencies in seismic analysis by comparing numerical results with experimental one for different shaped tank geometries.

In the present study, liquid sloshing behavior under harmonic excitation inside partially filled cylindrical tank is studied using commercial CFD software suite Fluidyn MP. Two cases with harmonic excitations at 5.16 rad/s and 5.74 rad/s are simulated. Fluidyn MP software suite is general purpose Multiphysics simulation software suite which includes “Fluidyn CAE” as geometry modeling tool, “Fluidyn MP” as grid generation, case setup and post processing tool and “Fluidyn MP Solver” as solver.

## 2. GEOMETRY MODELING

Cylindrical tank geometry is modeled using commercial software Fluidyn CAE. Diameter of the tank is considered as 60cm while height is taken as 20 cm. Modeled geometry is shown in fig.1.

Initial liquid and gas heights are modeled in computational setup itself. Mesh generation is done considering initial heights of different fluids.

Fluidyn CAE has capability of surface mesh generation and post processing also along with geometry modeling.

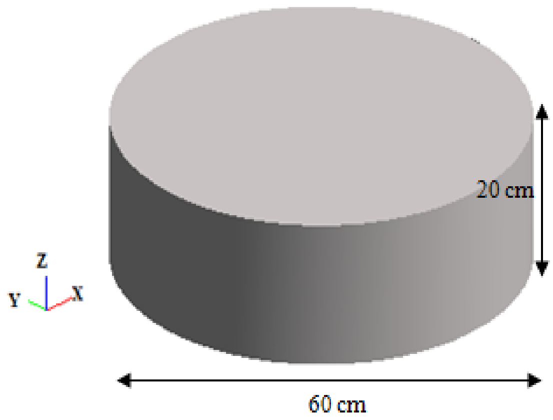



**Fig -1**: Cylindrical Tank

## 3. MESH GENERATION

The mesh for present study is generated by using cadgen utility of commercial software Fluidyn MP. Both block structured and unstructured mesh generations are possible with Fluidyn MP. For present study structured mesh with hexahedral elements is generated. Fine mesh is generated near the interface as well as near to the tank wall. 0.5 mm size is maintained near the interface. Details of the mesh are given in following table.

**Table -1:** Mesh Details

| | |
|---|---|
| Mesh Type | Structured Mesh |
| Element type | Hexahedral |
| No. of elements | 216000 |
| No. of nodes | 222131 |

Generated mesh/grid is shown in following figures

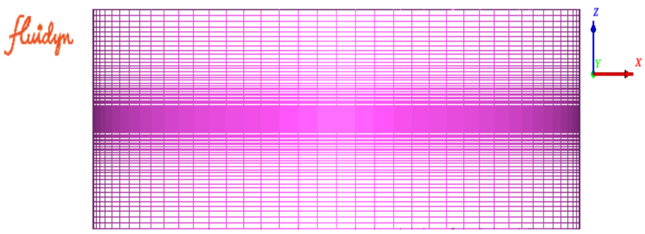


**Fig -2 (a)**: Computational grid, XZ view

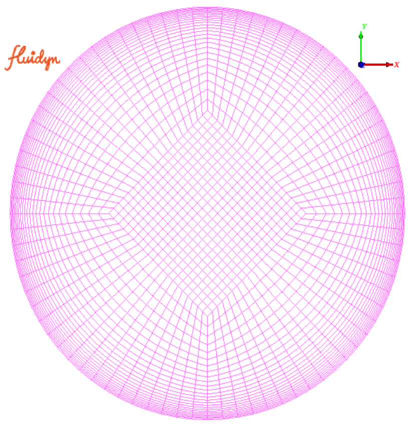


**Fig -2 (b)**: Computational grid, XY view

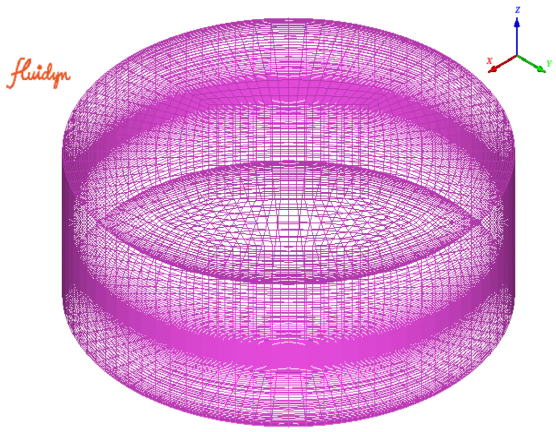


**Fig -2 (c)**: Computational grid, isometric view

## 4. BOUNDARY AND INITIAL CONDITIONS

Detailed boundary and initial conditions are shown in fig. 4. Pressure static boundary condition with 100000 Pa pressure is defined at the top surface of the tank, which is acting as the open surface. All other boundary surfaces (bottom and side) are defined with no slip wall boundary condition. Initially cylindrical tank is filled with water up to half of its height and other half is filled with the air. Initially entire domain is maintained at 100000 Pa pressure and 300K temperature condition.

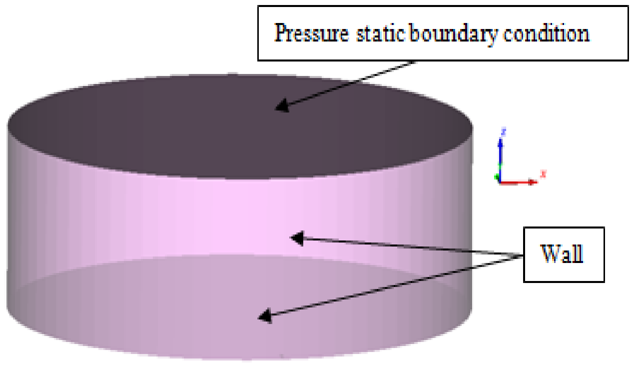



**Fig -3**: Boundary conditions

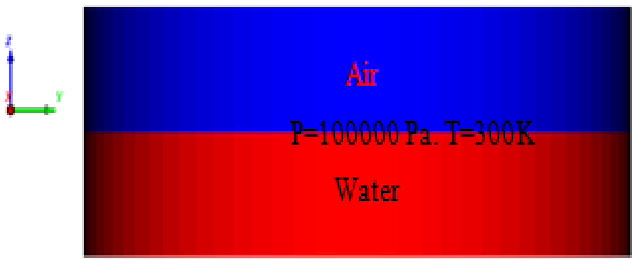



**Fig -4**: Initial Conditions

## 5. COMPUTATIONAL MODELING

Pressure based fully implicit transient solver is used for the present study. Flow is considered as laminar and gravity force is considered using full gravity model. Both Air and Water are considered as incompressible fluids with standard material properties for given pressure and temperature.

Pressure velocity coupling is carried out using SIMPLE algorithm. Free surface flow is modeled using volume of fluid (VOF) multiphase model while interface is captured using high resolution interface capturing scheme (HRIC). Both surface tension and wall adhesion effects are considered. Fluidyn-MP solves a convective equation for the volume fraction of the 2nd phase, the solution of which provides information on the position and shape of the interface between the phases. The equation for the volume fraction is:

$$\partial\phi/\partial t + \nabla.(\phi U) = 0$$

which implies that ϕ changes solely under the influence of convection. The variable ϕ has initial values of zero in one fluid (say air) and unity in the other (say water). Time step size is selected as 0.005 seconds and simulations are run up to 15 seconds.

Moving mesh modeling approach is used for considering the effect of harmonic excitation/oscillation. All the nodes of mesh are assigned with same X component of velocity calculated based upon lateral displacement equation:

$$x(t) = A\sin(\omega t)$$

Where, A=0.05 cm (A=Amplitude of motion in cm) and $\omega$=5.16 rad/s and 5.74 rad/s ($\omega$=excitation frequency in rad/s)

## 6. RESULTS

This section involves discussion about main obtained results of the given study. Result analysis is carried out using commercial software Fluidyn MP. Simulations are run for two different excitation frequencies 5.16 rad/s and 5.74 rad/s. Numerical results in terms of sloshing or wave height measured at right end of the computational domain are compared with experimental results given by Yung - Hsiang Chen et.al.[1]. Sloshing or wave height is measured using volume integral method. Figures 5 and 6 indicate the comparison of computational and experimental sloshing heights for the frequencies 5.16 rad/s and 5.74 rad/s respectively. Computational results from both the comparisons are found to be following the same trend as that of experimental results and observed to be qualitatively predicting the proper sloshing height. Considering all the limitations and approximations involved during the experimentation these computational results can be considered as satisfactory one [2].

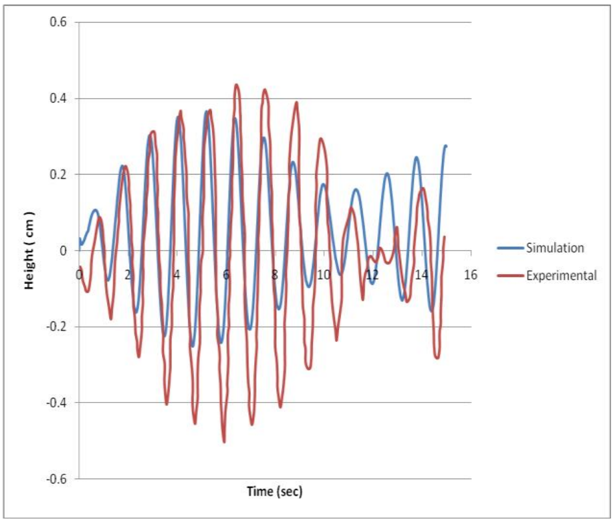



**Fig -5**: Sloshing height comparison, 5.16 rad/s

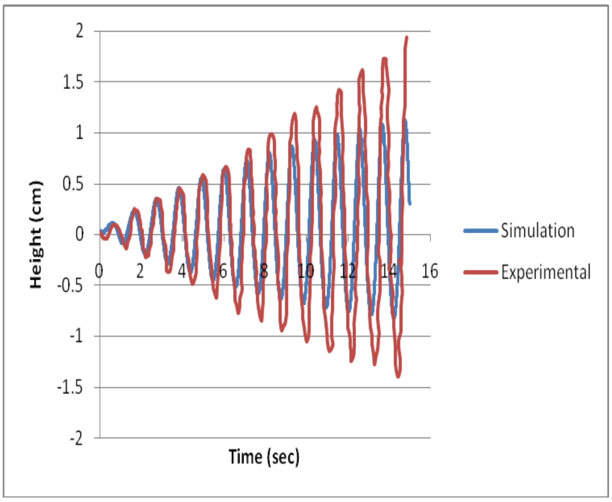
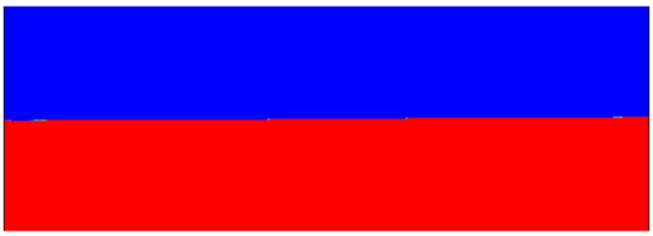



**Fig -6**: Sloshing height comparison, 5.74 rad/s

From figure 6, it is also observed that sloshing becomes violent and divergent at large excitation frequency resulting into computationally under prediction of sloshing heights. This can be due to the consideration of non- linear effects from potential equations [6].

Figures 7 shows the water volume fraction contours after 5 seconds for both the excitation frequencies 5.16 rad/s and 5.74 rad/s



(a)

(b)

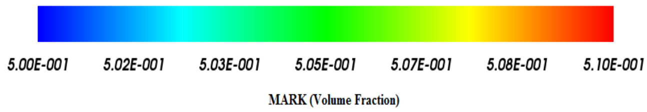
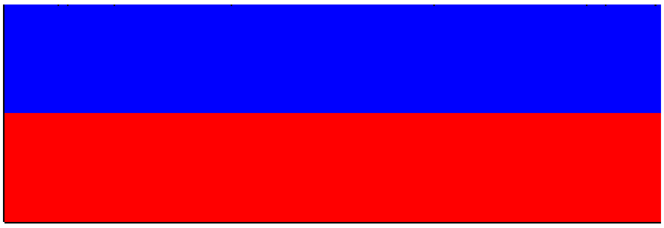
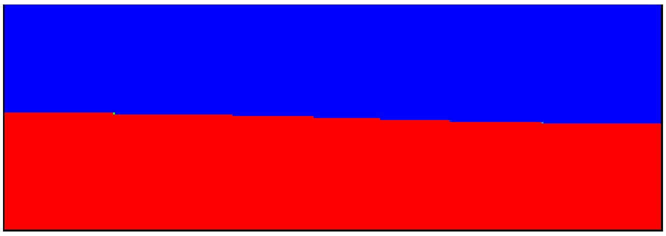



**Fig -7**: Water (MARK) volume fraction contours (a) 5.16 rad/s (b) 5.74 rad/s, after 5 sec

Figures 8 shows the water volume fraction contours after 10 seconds for both the excitation frequencies 5.16 rad/s and 5.74 rad/s



(a)



(b)

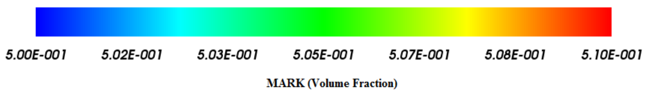



**Fig -8**: Water (MARK) volume fraction contours (a) 5.16 rad/s (b) 5.74 rad/s, after 10 sec

From figures 7 and 8 it is observed that increase in excitation frequency has increased the sloshing height and hence results into increase of pressure over the tank surface.

Increase in pressure is shown in figure 9. Pressure values for both the excitation frequencies are compared at the right end of the computational domain and same is shown in figure 9.

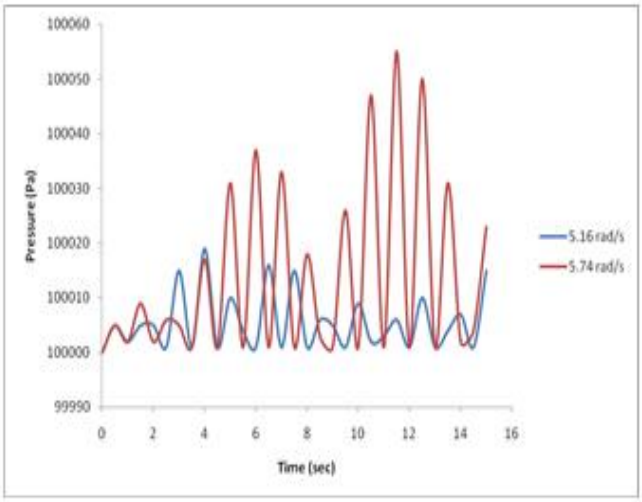



**Fig -9**: Pressure value comparison

## 7. CONCLUSION

In the present study, CFD analysis of a liquid sloshing behavior inside partially filled cylindrical tank at two different harmonic excitation frequencies is carried out using Fluidyn MP Software suite.

This study is carried out to understand the role of CFD to enhance safety in the design of storage tanks containing fluids.

In order to carry out CFD analysis finite volume cylindrical tank having diameter of 60 cm and height 20 cm is modeled.

Computationally obtained results in terms of sloshing height are observed to be in good agreement with experimental results. Considering the limitations and approximations involved in experimentation, obtained results can be considered as satisfactory one. Following conclusions are carried out from the present study.

1. VOF multiphase model is suitable to predict the tank sloshing accurately.
2. The results obtained by simulation shows the nonlinear behavior of liquid under intensive harmonic excitations with reasonable accuracy.
3. Simulation provides the reliable results which can be used for the design of liquid storage tank as sloshing depends on geometry of tank.
4. Increase in excitation frequency results into increase of pressure which is the main concern from tank safety point of view
5. Sloshing and its impact on the structure of the tank can be reduced by providing some shock absorbing devices which would damp oscillation in structure for their protection.

## ACKNOWLEDGEMENT

This study is supported by Fluidyn Consultancy Pvt. Ltd. Bangalore in terms of software support and research guidelines.

Authors extend their sincere thanks to Multiphysics solver development Department, Fluidyn for valuable guidance.

Authors also extend their sincere thanks to Aerospace Department, UPES, Dehradun for the valuable opportunity.

## REFERENCES

[1]. Yung - Hsiang Chen, Wei - Shien Hwang, Chia - Hao Ko: Sloshing behaviours of rectangular and cylindrical liquid tanks subjected to harmonic and seismic excitations. Earthquake Engineering and Structural Dynamics 2007; 36:1701-1717

[2]. P. Pal., "Sloshing of liquid in partially filled contained- An experimental study", International Journal of Recent Trends in Engineering, Vol. 1, No. 6, May 2009

[3]. Zeineb Saudi et.al, "Standing wave induced by free liquid sloshing in rectangular tank," International Renewable Energy Congress, Sousse, Tunisia, November 5-7, 2010.

[4]. Chung-Yueh Wang et. al., "Numerical simulation of sloshing motion inside a two dimensional rectangular tank with baffle or baffles". Journal of Aeronautics, Astronautics and Aviation, Series A, Vol.43, No.3 pp.207 - 216 (2010).

[5]. O. R. Jaiswal et. al., ".Study on Sloshing Frequencies of Fluid tank System", The 14th World Conference on Earthquake Engineering, 2008.

[6]. Neptune Yu et. al., "Nonlinear sloshing simulations under dynamic excitations", Altair Technology Conference Paper, 2014.