A NUMERICAL SLOSHING ANALYSIS FOR ASSESSMENT OF LNG FUEL TANK USING OPENFOAM®

HOTAK OK[1], JIWON CHOI[2], KWANGMIN LEE[3], JINHO YANG[4]
[1]*Hyundai Heavy Industries Co., Ltd., okhotak@hhi.co.kr*
[2]*University of Ulsan, zenith0804@ulsan.ac.kr*
[3]*Hyundai Heavy Industries Co., Ltd., kwangminlee@hhi.co.kr*
[4]*Hyundai Heavy Industries Co., Ltd., aceyang@hhi.co.kr*

Keywords: *InterDyMFoam, IsoAdvector, LNG Fuel Tank, Sloshing, VOF*

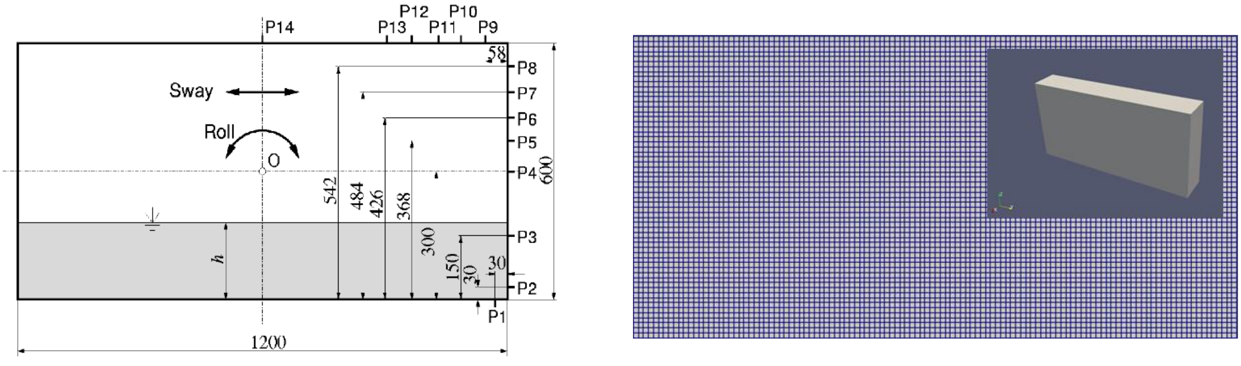
1. Introduction

The IMO (International Maritime Organization) has been promoting the reduction of pollutant emissions such as carbon dioxides, sulphur oxides and nitric oxides. And the environmental regulations are continuously strengthening and that makes the paradigm shift in the shipbuilding industry. Especially, there has been significant interest in the 2020 sulphur cap which means that ships will be banned from using fuel with sulphur content above 0.5% from January 1, 2020. As some shipping industries have decided to use LNG as fuel to satisfy the regulation, the needs for the LNG fuelled ships are increasing. However, in the fuel tank for LNG, the filling level changes during the ship's operation and the sloshing can be stronger at the specific filling level. As a result, the sloshing causes loads continuously on the wall of tank during the ship's lifetime.

There are several types for LNG tank and many shipowners choose the membrane type for cost effect. But the membrane tank is structurally weak because stiffeners are not installed enough to reduce costs. So, the sloshing tank is assessed strictly. For the assessment of sloshing tank, the CFD analysis is useful tool at the stage of initial tank design because experiment takes a lot of time and cost. However, there are still many difficulties in the numerical simulations since the sloshing has highly unsteady and chaotic behavior. And the effects of spatial and temporal discretization on pressure prediction are not clear. So, in the present study, we investigated numerical methods based on various versions of the OpenFOAM® to find practical procedure for sloshing CFD analysis.

2. Validation of Numerical Method

We first tested the feasibility of the current methods for the sloshing problem by comparing the predicted results with the data measured by Hinatsu[1]. The validation case has the water level of 20% and oscillating rolling motion with period 1.85 sec and 10-degree amplitude. The mesh was generated by using the blockMesh utility to form cells as close to the cube as possible. The computational domain consisted of 144,000 cells. Figure 1 shows the experimental set up and mesh arrangement.



(a) Experimental setup (b) Mesh arrangement

Figure 1: Experimental setup (left) and mesh arrangement for numerical simulations (right)

2.1. Numerical Method

We used the 2nd order schemes for spatial discretization and the Euler scheme was used for temporal discretization with adjustTimeStep. It is important to predict the free surface precisely and efficiently in sloshing simulations. To do this, we first used interDyMFoam, an incompressible interface capturing solver using VOF with dynamic mesh, which is basically provided in the OpenFOAM® where the 2.1.1 version was tested. We also employed OpenFOAM-v1712, supported by ESI-openCFD group, to test the latest VOF scheme, isoAdvector[2] for sloshing problems. Since the OpenFOAM-v1712 does not provide on interface capturing solver with dynamic mesh using the isoAdvector yet, we created one with dynamic mesh, so called interIsoDyMFoam. Figure 2 shows predicted and measured pressure over time at the sensor P2. We can see that the present methods well predicted the periodic pressure change at corresponding position. However the 2.1.1 version tended to under-estimated the peak pressure while v1712 overly predicted. It is worthy to note it is not possible to directly compare the predicted and measured pressures in numerical simulations and experiments respectively, but statistical properties of the pressure changes are much more important information in the assessment of tank sloshing.

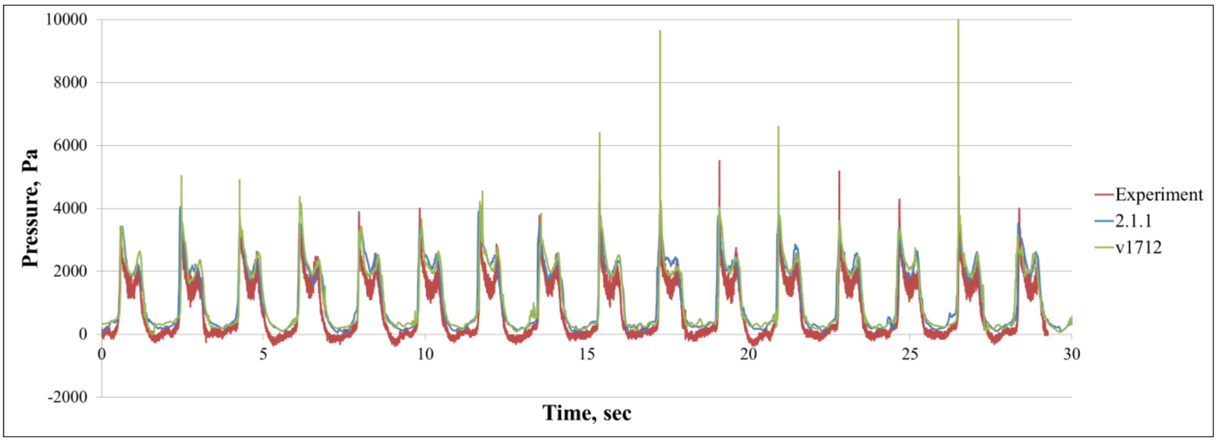


Figure 2: Comparison of predicted and measured pressure at sensor P2

We found some interesting points in a long term simulation. Figure 3 compares the predicted pressures from two different versions of the OpenFOAM. As compared, the abnormal pressure was predicted from 100 seconds. We are currently working on this part and will be presenting the results of research at the OpenFOAM Workshop.

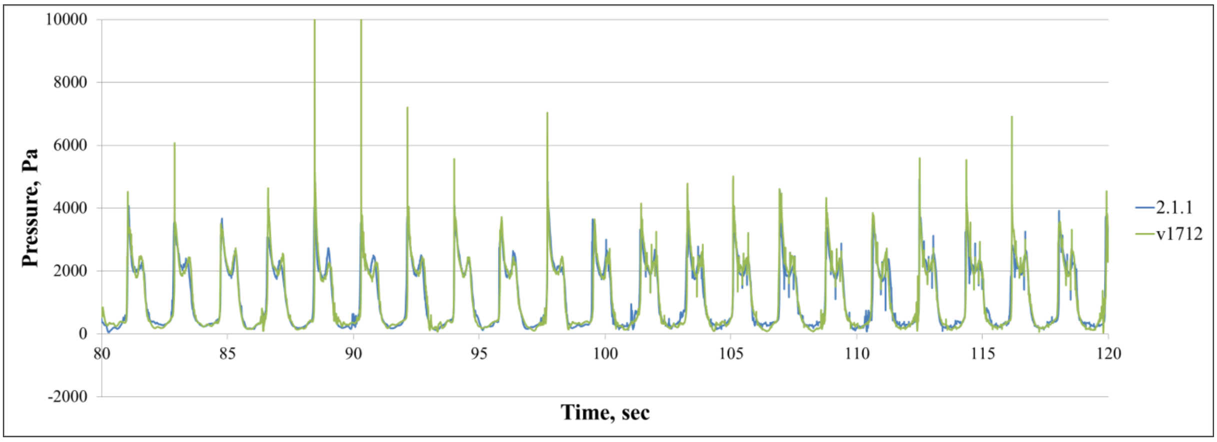


Figure 3: Comparison of predicted pressures near 100 seconds at sensor P2

2.2. Parameter Study

We compared the results with various values of the adjustTimeStep. The adjustTimeStep option adjusts the time step size to ensure the CFL number to be less than the specified value at every time step. Here, we can control the values of 'Max CFL' and 'Max Alpha CFL'. So, we compared the results by changing these two parameters. And the sloshing analysis is ambiguous to determine whether the flow itself is laminar or turbulent. We assessed the effect of the turbulence model as well. Two turbulence models were used, k-epsilon model and k-omega SST model. We will cover this study in presentation.

3. Numerical Assessment of LNG Tank

We selected a tank considered to be safe as the reference and performed an analysis on a new tank shape numerically and experimentally. Then we compare the results with the reference tank to see if the new tank will be safe. In the numerical investigation, the Aquarius has been used and the safety of the sloshing tanks has been assessed is evaluated by means of a statistical method [3]. In the present study, we compared the statistical properties obtained from the OpenFOAM-2.1.1 and v1712 to those of Aquarius. We have selected a tank of 150K LNG Carrier as the reference tank that has been operated safely for 10 years and assessed the safety of a tank of 177K LNG Carrier. Please note that the tank shapes could not be presented due to security regulation.

Simulations were carried out on a 2D tank in model scale over the physical time corresponding to 30 hours in the real scale. We tracked the peak pressures after the simulation and compared the average of the three top peak pressures and the maximum value of the peak pressures to the reference tank. The results are shown in Table 1. As we can see, the results from 2.1.1 showed similar values and tendency to the Aquarius while the v1712 predicted overly estimated results relative to the Aquarius.

Table 1: Pressure ratio of computational results against 150K LNG Tank

Tank	Filling Level	Aquarius		2.1.1		v1712	
		Max.	Top 3 Ave.	Max.	Top 3 Ave.	Max.	Top 3 Ave.
177K LNG Tank	All	1.06	1.13	1.01	1.03	1.43	1.23
	2.75 m	0.97	0.87	0.83	0.95	0.67	0.74
	70% H	1.06	1.13	1.33	1.16	1.43	1.23
	80% H	0.75	0.93	1.01	0.98	0.62	0.84
	95% H	1.00	0.74	1.15	0.98	1.01	1.03

4. Automation System

It is a time consuming task to create a new simulation case using the OpenFOAM® . In the sloshing problems, users may have difficulties in inputting mandatory data such as filling level, motion, and locations of pressure sensors, etc. Since more than 200 cases are required to assess a LNG tank and the resultant pressure data is too huge to be directly interpreted, it will be helpful to get an automatic post-processing code. We are developing an automation program for sloshing CFD analysis. Although this is not accomplished yet, we can share some information about the automations system in the workshop.

5. Conclusion

Various OpenFOAM versions have been tested to see their feasibility for assessing sloshing tanks. The VOF solvers provided by the OpenFOAM predicted reasonable solutions of sloshing problems when they were compared to the experimental data. Due to the extremely transient and chaotic nature of the sloshing problem, the pressure predicted in a numerical simulation cannot be compared directly with the measured experimental data. Instead, we compared statistical characteristics of a tank to the reference tank. The numerical simulations for the sloshing tank assessment were compared to those of the Aquarius of Lloyd's Register, which has long been used for sloshing simulations in HHI. The 2.1.1 showed comparative solutions to the Aquarius while v1712 with isoAdvector showed quite overly estimated results. We will conduct more strict numerical tests to verify the present numerical assessment procedure of a sloshing tank and it will be discussed in the presentation.

Acknowledgements

This research was carried out with the support of the technical development project of Hyundai Maritime Research Institute (HMRI) of Hyundai Heavy Industries Co., Ltd. We thank Dr. Geonhong Kim in HMRI for assistance with particular technique, and Prof. Sang-Wook Lee and Prof. Kyoungsik Chang from Ulsan university for comments that greatly improved the research.

References

[1] J. Roenby, H. Bredmose, and H. Jasak: A Computational Method for Sharp Interface Advection, Royal Society Open Science, Vol. 3, No. 11, Nov. 2016. DOI: 10.1098/rsos.160405, 2016

[2] M. Hinatsu: Experiments of two-phase flows for the joint research, in Proceedings of SRI-TUHH mini-Workshop on Numerical Simulation of Two-Phase Flows, National Maritime Research Institute & Technische Universitaet Hamburg, NMRI, 2001.

[3] Aquarius: a CFD-based software package, Lloyd's Register