Comparison of Three Commercial Software for Investigating Seismic Safety of the Spent Fuel Pool

Wonman Park [a], Sehong Oh [a], Dae Kyung Choi [a], Heung Seok Kang [b], Dong Seok Oh [b],
Kang Hee Lee [b], Choengryul Choi [a*]
[a]*ELSOLTEC, Giheung-gu, Yongin, Gyeonggi-do, 16950, Korea*
[b]*Korea Atomic Energy Research Institute, Yuseong-gu, Daejeon, 34057, Korea*
[*]*Corresponding author: crchoi@elsoltec.com*

1. Introduction

Earthquake causes sloshing of the coolant in the spent fuel pool (SFP). The sloshing could be the reason of the reduction of the coolant by overflow, also be the reason of the heavy load not only on the fuel assemblies but also on the SFP. Therefore, accurate investigation of the sloshing is critical for evaluating seismic safety of the SFP.

Ansys Fluent, Ansys CFX (Ansys Inc., Canonsburg, PA, USA), and STAR-CCM+ (CD-Adapco, Melville, NY, USA) are the most widely used codes in the nuclear safety community [1]. Theses commercial computational fluid dynamics(CFD) software can be used for accurate investigation of the sloshing in the spent fuel pool. However, the results of sloshing analysis using different CFD software have not been compared in the previous studies. In this study, sloshing of the water in the scaled-down spent fuel pool was analyzed using three-different CFD commercial software and the results were compared with each other to identify similarity and difference of the results of CFD software.

2. Methods and Results

A rectangular tank (height = 0.815 m, width = 0.52 m, and depth = 0.15 m) which is designed based on the real spent fuel pool at South Korea's Hanul nuclear power plant was considered in this study (Fig. 1). Volume of fluid models including filled water and air in the tank were developed for three commercial CFD software (Ansys Fluent, Ansys CFX, and STAR-CCM+) to analyze water sloshing in the tank. Three different model which have initial level of the filled water of 0.23 m, 0.46 m, and 0.61 m respectively were considered. Static pressure at the pressure outlet, which is the superior plane of the calculation domain, was set to be 0 Pa. All the wall boundaries were treated as no-slip conditions. The temperature of the water and air was assumed to be 25 ℃ and those material properties were used accordingly. Thus, 1.185 kg/m^3 and 1.831e-5 kg/(m·s) was used for the density and viscosity of the air, and 997 kg/m^3 and 8.899e-4 kg/(m·s) was used for those of the water. Surface tension was set to be 0.07199 N/m.

The calculation domain was excited for 10 seconds with varying excitation frequency from 0.5 Hz to 10 Hz. Amplitude of 5 mm was used for all excitation frequencies. Changes in the pressure on the wide wall of the tank at the height of 120 mm, 235 mm, 615 mm, and 676 mm from the floor surface of the tank were measured for 15 seconds (10 seconds during excitation and 5 seconds during excitation). Then Fast Fourier Transform was employed to extract frequency content of pressure changes. Changes of free surface on the left-side tank wall were also measured during the same period. All the predicted results from three commercial CFD software was compared with each other.

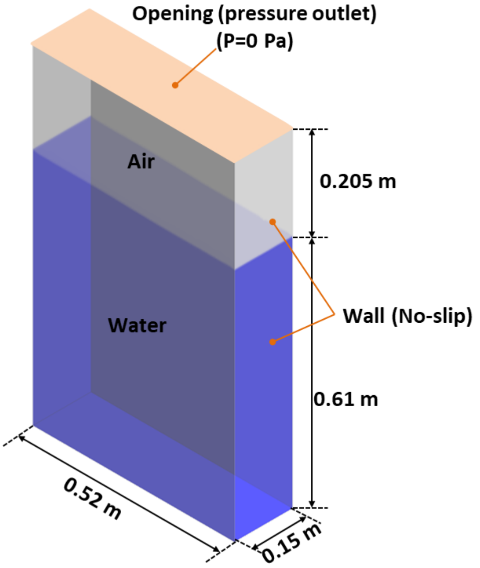


Fig. 1. Calculation domain and boundary conditions for sloshing analysis

3. Results and Discussion

While cycle of changes in wall pressure around tank floor (at the height of 120 mm and 235 mm) showed the same cycle of excitation 10 Hz, changes in the wall around free surface showed much lower cycle because of sloshing of the water (Fig. 1 and 2). These results could explain that rigid beam-mass and spring-mass were used according to depth of the filled water in equivalent model of the water [2]. The same patterns were shown in all software.

Frequency content of pressure changes around free surface was about 1.2 Hz, when initial water level was 610 mm (Fig. 3). This value is very close to the first mode natural frequency of water sloshing calculated using linear wave theory, and this phenomenon is well matched with the results from the previously published study [3].

Correlation of height/width of the water and sloshing frequency was calculated using linear wave theory and those were compared with computational results (Fig. 4). First five natural frequencies of the sloshing water were shown as solid lines in the Fig. 4. Here, h, L, f, and g are height of free surface, width of the tank, natural frequency of water sloshing, and gravity, respectively. While the second mode natural frequency extracted from computational results differed from the results of linear wave theory, the first mode natural frequency was well matched with the analytical results.

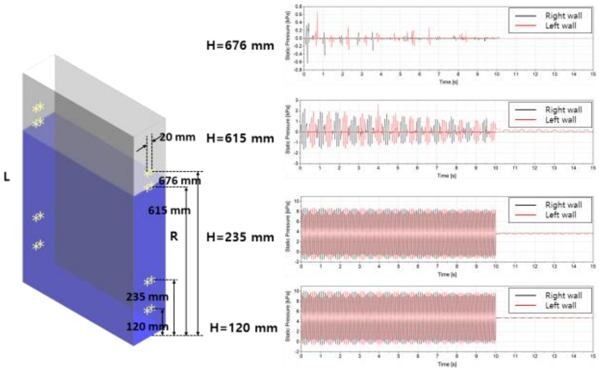
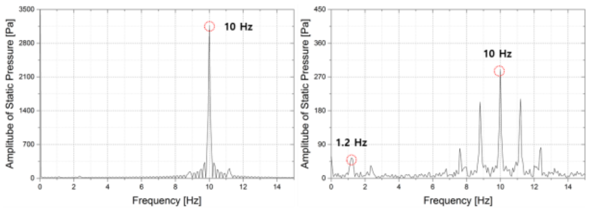


Fig. 2. Changes in the pressure on the tank wall predicted by Ansys Fluent, when excitation frequency and initial water level were 10 Hz and 610 mm, respectively.



H = 120 mm

H = 615 mm

Fig. 3. Frequency content of changes in wall pressure at the height of 120 mm (left) and 615 mm (right).

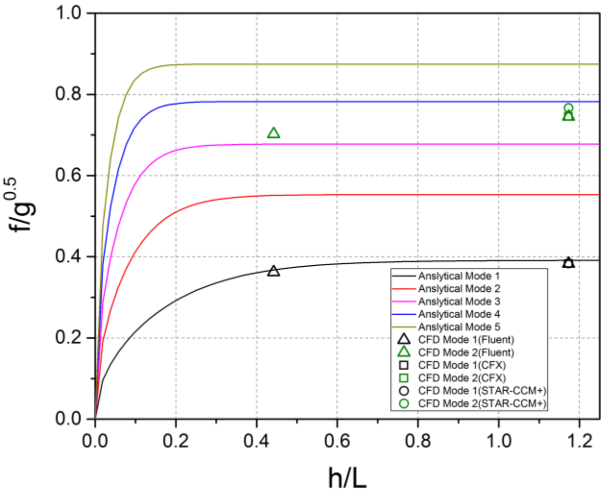


Fig. 4. CFD results of sloshing frequency compared to linear wave theory

Predicted changes in water level on the left tank wall from three CFD software were measured and compared with each other (Fig. 5). Overall trends of the changes were very similar with each other and only 25 mm of root mean square errors among results were calculated in the worst cases in which the excitation frequencies were 10 Hz.

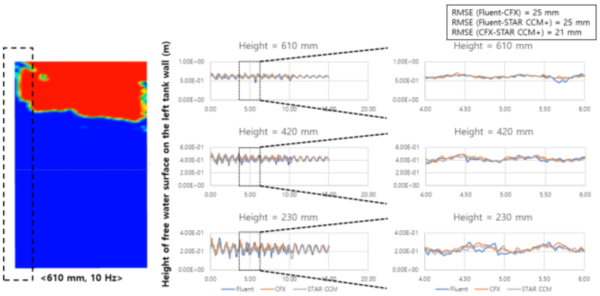


Fig. 5. Changes in water level on the left tank wall in cases that initial water levels were 610 mm, 420 mm, and 230 mm.

4. Conclusions

In this study, various sloshing phenomena were investigated using three commercial CFD software including Fluent, Star-CCM+, and CFX. Even though sloshing phenomena under only sine waves were considered in this study, the results of the three-different commercial software showed good agreement with each other. The results of this study could provide fundamental information to choose a commercial CFD software to investigate sloshing phenomena which affect the seismic safety of the spent fuel pool.

Acknowledgments

This work was supported by the Korea Institute of Energy Technology Evaluation and Planning(KETEP) and the Ministry of Trade, Industry & Energy(MOTIE) of the Republic of Korea (No. 20171510101920)

REFERENCES

[1] C. Boyd, Sr. Computational Fluid Dynamics for Nuclear Safety Analysis, NRC’s 24th Annual Regulatory Information Conference, Mar. 13-15, 2012, Rockville, MD.
[2] G. W. Housner, The Dynamic Behavior of Water Tanks, Bulletin of the Seismological Society of America, Vol. 53, P. 381, 1963
[3] H. Akyildiz, N. E. Unal, Sloshing in a three-dimensional rectangular tank: Numerical simulation and experimental validation, Ocean Engineering, Vol. 33, P. 2135, 2006