Transactions of the Korean Nuclear Society Autumn Meeting
Yeosu, Korea, October 25-26, 2018

Prediction of Fuel Storage Racks Motion Using Analytical and Numerical Methods for Seismic Assessment of Spent Fuel Pool

Sung Man Son [a], Dae Kyung Choi [a], Won Man Park [a], Se Hong Oh [a], Heung Seok Kang [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

It is critical to predict the rack motions subject to the seismic load in the seismic assessment of the spent fuel pool (SFP), because the collisions among racks or between racks and spent fuel pool might cause damages on the SFP, racks, and spent fuel assemblies. To predict the rack motions subject to the external excitation such like seismic force, several method, including analytical method, multibody-dynamic simulation, and finite element analysis could be used. The motions of the rack in the SFP is the resultant motion of the complex physics of the external load including seismic force, convection of the fluid filled in the SFP, and friction between racks and liner of the SFP. Thus, to estimate the rack motion subject to seismic load, the fluid structure interaction is necessary. Commercial finite element analysis software such as Ansys Mechanical (Ansys Inc., Canonsburg, PA, USA) and Abaqus (Simulia, Providence, RI, USA) could be used for estimating the rack motions, because they support fluid structure interaction.

However, it has not been reported how the commercial software predicts the rack motion subject to the external excitation, and different software might estimate different rack motions. In this study, as a first study to estimate rack motion in the SFP, the rack motions subject to several periodic external excitations were predicted with neglecting fluid effects using two commercial software Ansys Mechanical and Abaqus, and the results were compared with each other and with analytical results.

2. Analytical method

A custom Matlab code (Matlab 2018a, MathWorks, Inc., Natick, MA, USA) was developed to predict the rack motions analytically based on dynamic theory. The motion of the object rack could be estimate in two conditions which are slip and non-slip conditions. The calculation algorithm for the analytical solution was shown in Fig. 1. Here, g and ρ are gravitational acceleration and friction coefficient, respectively. Also, $d_{ground}(t)$, $v_{ground}(t)$, $a_{ground}(t)$, $d_{object}(t)$, $v_{object}(t)$, and $a_{object}(t)$ are displacement, velocity, and acceleration of the ground and object at time t, respectively.

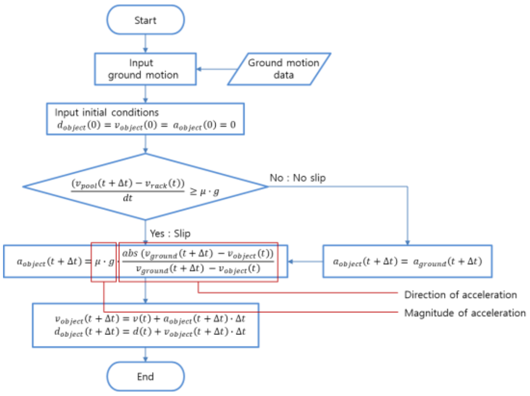


Fig. 1. Computational algorithm to predict rack motions during excitation

3. Numerical model and conditions

For the investigation of the nuclear power plant, only approved commercial software could be used. In this study, Ansys Mechanical and Abaqus were chosen, because these are approved.

The finite element model consists of two part, which are the liner of the SFP and a rack. Both parts were simplified as cubic shapes (Fig. 2). The length, width, and height of the liner was set to be 200 mm, 100 mm, and 5 mm, respectively. Those of the rack were set to be 100 mm, 50 mm, and 50 mm, respectively.

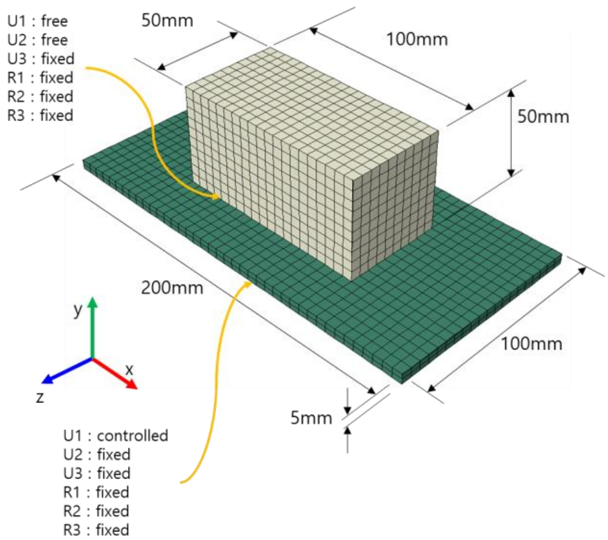


Fig. 2 Simple finite element of the rack and liner. Boundary conditions were also presented in the model.

Material properties of the stainless steel (E=200 GPa, ν=0.3, ρ=8,000 kg/m^3) were applied for both rack and liner. The three-dimensional surface-to-surface contact condition between the bottom plane of the rack and the upper plane of the liner was applied with friction coefficient of 0.3. Penalty method was adopted. For the boundary conditions, all motions except translation along x and z directions were constrained for the rack to prevent abnormal motion and allow effects of the slip and gravity. The motions of the liner were controlled in the x direction to simulate pre-defined excitations which are 1 Hz, 10 Hz, and 25 Hz excitations with amplitudes of 20 mm, 2 mm, and 2.5 mm, respectively (Table 1).

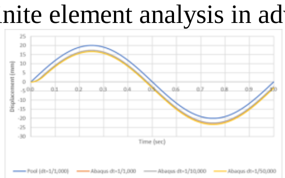
Table 1 Testing conditions

Case #	Amplitude	Frequency	Shape	Time(Δt)
	mm	Hz		s
1	20	1	Sine	1/1,000
2	2	10	Cosine	1/10,000
3	2.5	25	Cosine	1/50,000

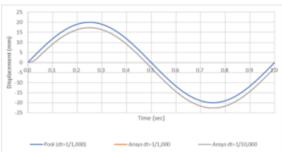
4. Results and Discussion

The predicted displacement of the rack using three methods were compared (Fig. 3-5). For excitation frequency 1 Hz, the results using all three methods showed good agreement with each other. Moreover, the results using different time increment also were identical.

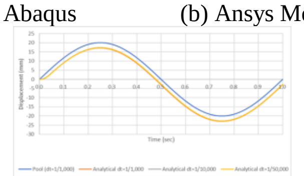
However, when excitation frequency increased to 10 Hz and 25 Hz, different motions of the rack were predicted with different time increment and different methods. Slip motion is continuously occurred in both 10 Hz and 25 Hz test conditions, while the slip is only occurred in the initial motion in 1 Hz test condition. Thus, different estimation of the slip motion should be the reason of the differences. In this study, penalty method was used for both Ansys Mechanical and Abaqus. In the method, penetration depth could affect the slip motion and different commercial software might predict different penetration depth. Therefore, validation process with comparing to the experiments should be necessary to estimate slip motion by using finite element analysis in advance.



(a) Abaqus

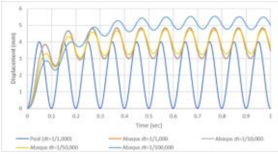


(b) Ansys Mechanical

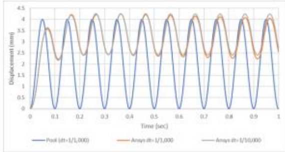


(c) Matlab

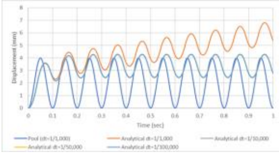
Fig. 3 Predicted rack motions in the excitation condition of 1 Hz, 20 mm



(a) Abaqus

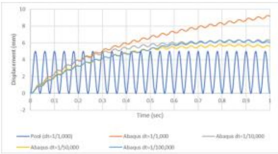


(b) Ansys Mechanical

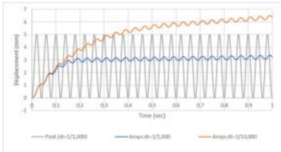


(c) Matlab

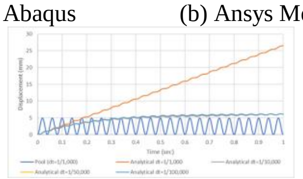
Fig. 4 Predicted rack motions in the excitation condition of 10 Hz, 2.0 mm



(a) Abaqus



(b) Ansys Mechanical



(c) Matlab

Fig. 5 Predicted rack motions in the excitation condition of 25 Hz, 2.5 mm

5. Conclusion

The results of this study showed that two commercial finite element software, Ansys Mechanical and Abaqus, predicted similar results with each other and with the analytical results in the low frequency testing condition. However, the commercial software predicted different results in the high frequency testing condition. Moreover, the different rack motions were predicted in all methods used in this study with varying time increment. Therefore, contact algorithm and time increment should be carefully selected in the prediction of slip motion of the rack.

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] ANSYS User's Guide R18.0
[2] Abaqus User's Manual 6.14