Progressive Calibration of the FE model of an Electric Cabinet in Nuclear Power Plant

Xuyen Thi Hong Nguyen [a], Tuan Thanh Tran [a], Dookie Kim [a*], Gihwan So [b]
[a]*Department of Civil Engineering, Kunsan National University, 558 Daehak-ro,Gunsan-si, Republic of Korea*
[b]*Innose Tech Co. LTD., 30 Songdomirae-ro, Yunsu-gu, Republic of Korea*
[*]*Corresponding author: kim2kie@kunsan.ac.kr*

1. Introduction

The electric cabinets are usually installed at some nuclear power plants situated in seismically active zone. Therefore, it is necessary to evaluate the nonlinear seismic analysis of these electronic cabinets. Many researches are conducted to expand knowledge on the dynamic performances of cabinet structures. The methods of simulation and experiment are presented in the works of many researchers [1-3].

The main aim of this work is to suggest a simplified numerical model for seismic analysis of electronic cabinet. The numerical model for cabinet is generated using the finite element approach with the Ansys software [4], which can reflect the realistic performances of the cabinet. The obtained modal analysis results are compared with the availability of experimental testing to validate the model.

2. Modal Simulation Analysis of Cabinet

2.1 Geometry of Cabinet

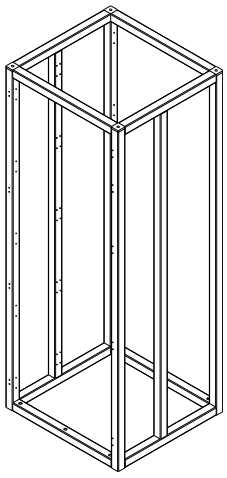

Fig. 1. The simplified model of cabinet

The cabinet dimensions are given as follows: length = 800 mm, width = 800 mm, height = 2100 mm as shown in Figure 1. All members of cabinet are assigned the steel material having the following properties: the modulus of elasticity, E = 2×10^{11} Pa, poisson's ratio, ν = 0.3, density, ρ = 7850 kg/m^3. The total weight of cabinet is 200kg. The fundamental frequencies in X- and Y-direction are 9.48Hz and 25.98Hz respectively.

2.2 Finite Element Model of Cabinet

The finite element model of electric cabinet using Ansys is shown in the Figure 2. The simplified cabinet model is developed using only beam element and without the cover plates. All elements are meshed with automatic method with size of 20 mm. The model of cabinet is simulated with the procedures of modelling, meshing and calculation.

The process of building the model is conducted by the following sections:

(1) Setting the job name
(2) Defining the material properties
(3) Creating the model
(4) Meshing and solving the model

The purposes of modal analysis are to determine the fundamental natural frequencies and mode shapes of electric cabinet that their results are displayed in the next chapter.

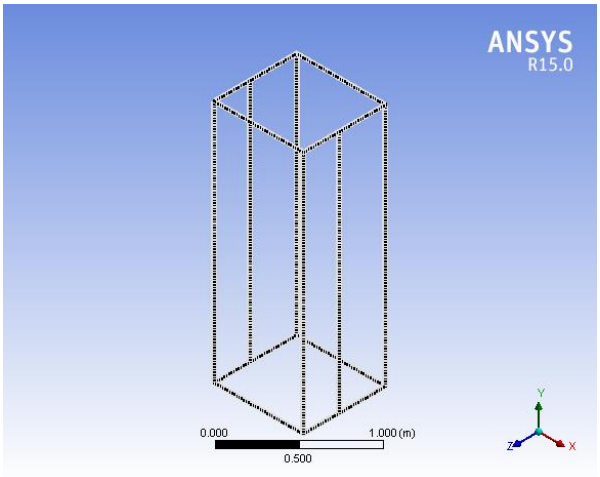

Fig. 2. The meshing of simplified model

3. Results and Discussions

Modal analysis, also known as free vibration analysis, is used to determine the natural frequencies and mode shapes of a structure. Modal analysis of electronic cabinet is performed to determine the

vibration characteristics, natural frequencies and mode shapes of cabinet. The natural frequencies obtained from the simulation model are compared with the experimental results. The natural frequencies and corresponding order natural frequencies are shown in Table I. The mode shapes of cabinet are shown in the Figure 3.

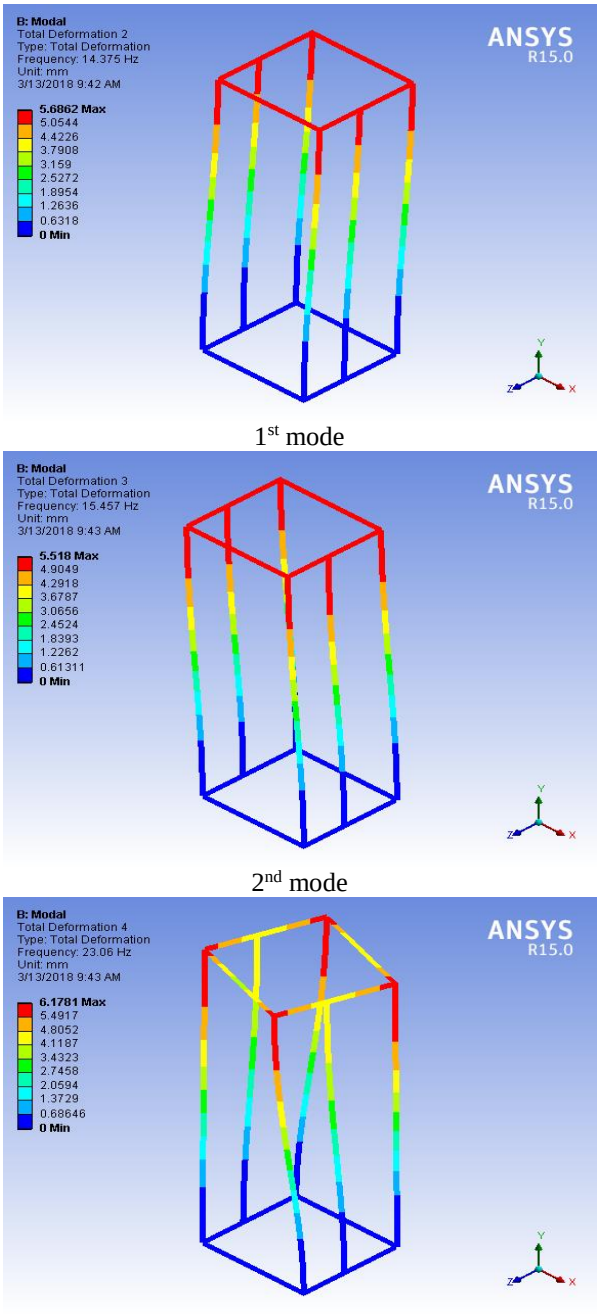

Fig. 3. The mode shapes of cabinet

Table I: The natural frequencies of model (Hz)

Mode	Frequency (Hz)
1	14.375
2	15.457
3	23.06

There are differences between the results of experimental and simulation model. This observation leads to conclusion that, the cover plates have significant effect on the numerical model.

The deformations of the first and second mode shape locate at the joint of top beam, the third one is torsion deformation of whole body. Through the analysis results, we can see that the deformation amplitude at each mode increases from the blue to red as shown in the Figure 3. The strongest amplitude is at the red part, while the deep blue is the weakest place.

4. Conclusion

In order to investigate the response of cabinet in nuclear power plant, a simplified model of structure is simulated with a finite element model that this is a very valuable tool in understanding a cabinet's structural behavior. The modal analysis is performed with the natural frequency of 14.375 Hz, sufficiently close to the experimentally measured frequency of 9.48 Hz. Although, Ansys simulations cannot exclude the experimentations, but greatly reduce the design of experiments where time and cost do not allow experimentations.

ACKNOWLEDGMENT

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. 20171510101960)

REFERENCES

[1] Katona, T., Kennerknecht, H., & Henkel, F. O. (1995). Earthquake design of switchgear cabinets of the VVER-440/213 at Paks. In Transactions of the 13. International Conference on Structural Mechanics in Reactor Technology. v. 3.

[2] Cho, S. G., Cui, J., & Kim, D. (2011). A procedure for generating in-cabinet response spectra based on state-space model identification by impact testing. Nuclear Engineering and Technology, 43(6), 573-582.

[3] Koo, K. Y., Cho, S. G., Cui, J., & Kim, D. (2010). Seismic response prediction for cabinets of nuclear power plants by using impact hammer test. Nuclear Engineering and Design, 240(10), 2500-2511.

[4] ANSYS, A. (2013). Version 15.0; ANSYS. Inc.: Canonsburg, PA, USA November, 752.