Transactions of the Korean Nuclear Society Spring Meeting

Jeju, Korea, May 18-19, 2023

Development of Automated Structural Analysis Platform for Data-Driven Prediction on Structural Responses

Jingoo Lee [a], Seungjun Lee [a], Young-Joo Lee [a*]
[a]*Dept. of Urban and Environmental Engr., Ulsan National Institute of Science and Technology (UNIST), 50 UNIST-gil, Eonyang-eup, Ulju-gun, Ulsan 44919, Rep. of Korea.*
[*]*Corresponding author: ylee@unist.ac.kr*

1. Introduction

As the failure of nuclear power plants subject to earthquakes can cause enormous economic and social losses, their structural safety is considered important [1]. Conventionally, the safety evaluation and response prediction of a structure subject to earthquakes requires introducing a physics-based-model (e.g., finite element (FE) model) [2-4]. However, a conventional approach may have limitations, particularly when the introduced physics-based-model is complicated and needs to be analyzed for a large number of uncertainty factors and earthquake ground motions, because such a structural dynamic analysis using a sophisticated FE model can be computationally expensive.

To overcome the research challenge, in the recent years, data-driven approaches have been applied extensively [5]. For the data-driven prediction on the response of a complicated structure, the structural responses of a target structural system for various values of random variables and several earthquake excitations need to be collected first [6,7], which requires repeated FE simulations. However, it is often impractical to perform a large number of FE simulations while changing the FE model parameters manually.

This paper presents a new computational platform, Monte Carlo ABAQUS, which enables automated FE analyses. Among various software packages of structural analysis, ABAQUS [8,9] was selected because it is widely used for various structures including nuclear power plants. The automated structural analysis platform can generate an expected number of random samples based on Monte Carlo simulation (MCS). Then, the corresponding input files are generated, and FE simulations using ABAQUS are performed. Finally, the analysis results are collected. Monte Carlo ABAQUS automates all these processes, making it easy to collect structural response data for the response prediction of a structure.

2. Developed Platform: Monte Carlo ABAQUS

Monte Carlo ABAQUS is the combination of MCS, which is a widely-used sampling-based method, and ABAQUS. The overall process of Monte Carlo ABAQUS consists of three steps: (1) random sample generation, (2) ABAQUS input file modification, and (3) ABAQUS simulation and response collection. The operation process of Monte Carlo ABAQUS is briefly shown in Fig. 1.

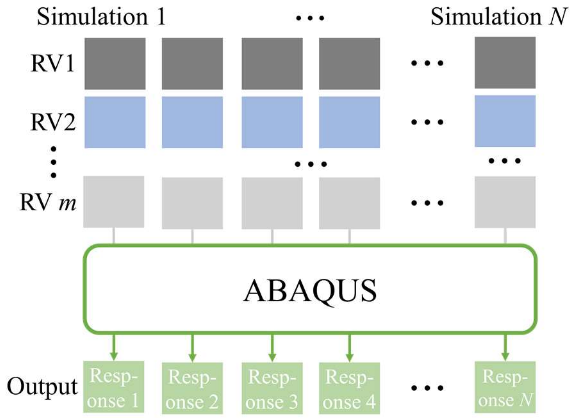

Fig. 1. Operation process of Monte Carlo ABAQUS

The first important step of MCS is typically random sample generation for random variables (RVs) which contain the uncertainty associated with a structure, and this is the same for Monte Carlo ABAQUS. Based on the prescribed statistical information about m RVs, Monte Carlo ABAQUS generates N samples sets. In this study, the developed computational platform was mainly constructed in MATLAB because it provides several built-in functions for random sample generation with various probabilistic distributions (such as normal, lognormal, exponential, and Gumbell distributions).

Once random sample sets are generated, the original ABAQUS input file (i.e., in the "*.inp" file format) is automatically modified to have the corresponding random sample values. For the task, special marking techniques are applied to the original ABAQUS input file, which allows MATLAB to recognize the locations and original values of the RVs that need to be modified.

Then, ABAQUS simulations are performed using the modified input files. ABAQUS analysis can be ordered by using commands in the DOS mode, and MATLAB enables this ABAQUS control. In addition, Monte Carlo ABAQUS contains a Python-based interface code which enables the access to "*.odb" file, the output database of ABAQUS. Among the various structural responses accessible within the output file such as displacement, force, and stress, Monte Carlo ABAQUS selectively extract and stores only the desired kinds of data. Techniques similar to those mentioned above have been applied in previous studies [10-12], and these techniques were modified so that they can be applied to MCS and used in this computational platform. Eventually, through this process, N structural responses can be automatically obtained for N random sample sets.

3. Illustrative Example

To illustrate the developed computational platform, it is applied to a simple hypothetical wall structure. The ABAQUS model of the example structure is shown in Fig. 2. As shown in the figure, it is assumed that the wall is fixed to the ground (i.e., bottom), and it is subjected to a uniform transverse load.

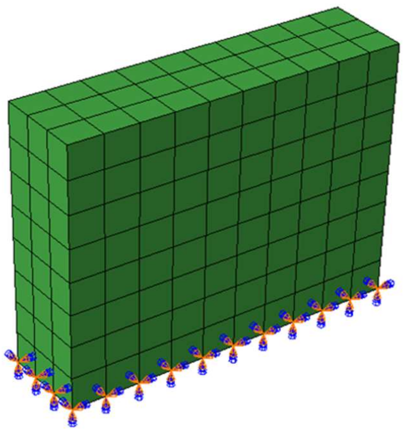

Fig. 2. ABAQUS model of illustrative example

In this example, there are assumed to be three RVs about density, elastic modulus, and load. One of the key techniques of Monte Carlo ABAQUS is to automatically modify the original ABAQUS input file in accordance with the generated random sample sets of RVs. As described in Sec. 2, Monte Carlo ABAQUS introduces special marking techniques, and Fig. 3 shows a part of the original and modified ABAQUS input files.

```
** MATERIALS
**
*Material, name=Material-1
*Density
500.,
*Elastic
1.9e+08, 0.3
**
** LOADS
**
** Name: Load-1   Type: Pressu
*Dsload
Surf-1, P, 1000.
```

(a)

```
** MATERIALS
**
*Material, name=Material-1
*Density
% 1 % 500.%,
*Elastic
% 2 % 1.9e+08 %, 0.3
**
** LOADS
**
** Name: Load-1   Type: Pressu
*Dsload
Surf-1, P,% 3 % 1000. %
```

(b)

Fig. 3. (a) Original and (b) modified ABAQUS input files

First, the number of each RV is placed between two % symbols. At this time, the RV number must be the same as the one used when generating random sample sets. Next, the value that needs to be modified is enclosed with the following % symbol. In this way, Monte Carlo ABAQUS can recognize the locations and original values of RVs and modify the ABAQUS input files.

To test the computational platform, in this example, 1,000 samples are generated for each RV, which means a total of 3,000 random numbers for all RVs (i.e., m=3, N=1,000). In addition, the maximum displacement on top is extracted from the output file after each FE simulation, and a total of 1,000 top displacements are automatically collected. Fig. 4 shows the histogram of these values. Although Monte Carlo ABAQUS is applied to a simple structure in this study, it will be applied to more complicated structures in an actual nuclear power plant subject to earthquakes. It is expected that the collected data allow us to predict the seismic response and detect the anomaly of the target structure efficiently.

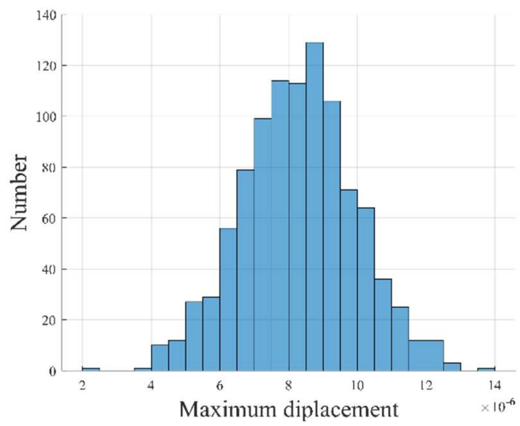

Fig. 4. Histogram of maximum displacements on top

In addition to such a structural response dataset, as a byproduct, Monte Carlo ABAQUS can provide the failure probability of a structure for a given limit state, as in a typical MCS. For example, in this study, the limit-state function $g(\mathbf{x})$ is assumed to be as follows:

$$g(\mathbf{x}) = 1.0\times10^{-5} - d(\mathbf{x}) \qquad (1)$$

where $\mathbf{x}$ is the vector of random variables and d is the top displacement. Typically, $g(\mathbf{x})\leq 0$ represents the failure of a structure in reliability analysis. In this example, it means that the wall structure fails when its top displacement exceeds 1.0×10^{-5}. Then, the failure probability can be obtained by dividing the number of failure cases by the total number of samples, and it is calculated as 0.153 (=15.3%) in this example. Fig. 5 shows how the probability obtained from MCS converges as the number of samples increases. The coefficient of variation of the failure probability result for 1,000 samples is estimated to be $[(1-0.153)/(1{,}000\times0.153)]^{0.5}=0.074$, which is at an acceptable level.

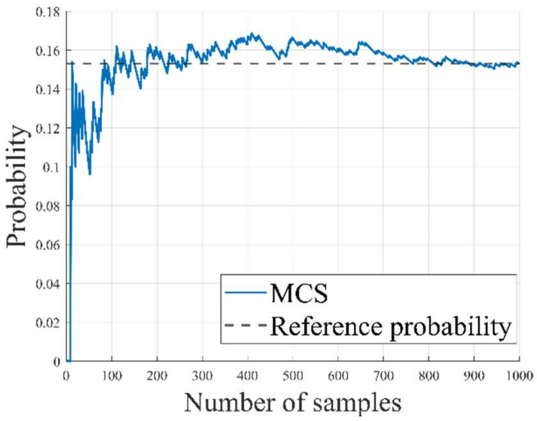

Fig. 5. Probabilities from MCS with increasing number of samples

4. Conclusions

In this study, a new automated computational platform of structural analysis, Monte Carlo ABAQUS, is developed for data-driven prediction on structural responses. The overall process of Monte Carlo ABAQUS can be summarized in three steps: (1) random sample generation, (2) ABAQUS input file modification, and (3) ABAQUS simulation and response collection. The developed platform was constructed in MATLAB, which makes it possible to control ABAQUS automatically. In addition, Monte Carlo ABAQUS introduces special marking techniques so that the locations and original values of RVs can be recognized. The computational platform is tested through its application to a simple hypothetical wall structure, and the structural response of interest (i.e., top displacements) is successfully obtained for 1,000 samples. It is also confirmed that the failure probability of a structure for a given limit-state function as a byproduct. Monte Carlo ABAQUS will be applied to more complicated structures in an actual nuclear power plant, and it is expected that the collected data allow us to predict the seismic response and detect the anomaly of the target structure efficiently.

ACKNOWLEDGEMENT

This work was supported by the National Research Foundation of Korea (NRF) grant funded by the Korea government (MSIT) (No. RS-2022-00144434).

REFERENCES

[1] Y. G. Zhao, M. J. Qin, Z. H. Lu, and L. W. Zhang. Seismic Fragility Analysis of Nuclear Power Plants Considering Structural Parameter Uncertainty. Reliability Engineering & System Safety, Vol. 216, No. 107970, pp. 1-12, 2021.

[2] S. F. Fu, H. Gao, Q. J. Zhang, and X. M. Zhang. Structural Displacement Response Prediction Based on Finite Element Method. Advanced Materials Research, Vol. 268, pp. 106-109, 2011.

[3] W. X. Ren and H. B. Chen. Finite Element Model Updating in Structural Dynamics by Using the Response Surface Method. Engineering Structures, Vol. 32, No. 8, pp. 2455-2465, 2010.

[4] K. J. Bathe, H. Zhang, and S. Ji. Finite Element Analysis of Fluid Flows Fully Coupled with Structural Interactions. Computers & Structures, Vol. 72, No. 1-3, pp. 1-16, 1999.

[5] G. Hu, T. Zhou, and Q. Liu. Data-Driven Machine Learning for Fault Detection and Diagnosis in Nuclear Power Plants: A Review. Frontiers in Energy Research, Vol. 9, No. 663296, pp. 1-12, 2021.

[6] B. K. Oh, B. Glisic, S. W. Park, and H. S. Park. Neural Network-Based Seismic Response Prediction Model for Building Structures Using Artificial Earthquakes. Journal of Sound and Vibration, Vol. 468, No. 115109, pp. 1-20, 2020.

[7] T. Li, Y. Pan, K. Tong, C. E. Ventura, and C. W. de Silva. A Multi-Scale Attention Neural Network for Sensor Location Selection and Nonlinear Structural Seismic Response Prediction. Computers & Structures, Vol. 248, No. 106507, pp. 1-17, 2021.

[8] G. Abaqus, Abaqus 6.11. Dassault Systemes Simulia Corporation, Providence, RI, USA, 2011.

[9] S. Raychaudhuri. Introduction to Monte Carlo Simulation. 2008 Winter simulation conference, pp. 91-100, 2008.

[10] J. Lee, Y.-J. Lee, H. Kim, S. Sim, and J. Kim. A New Methodology Development for Flood Fragility Curve Derivation Considering Structural Deterioration for Bridges. Smart Structures and Systems, Vol. 17, No. 1, pp. 149-165, 2016.

[11] H. Kim, S.-H. Sim, J. Lee, Y.-J. Lee, and J.-M. Kim. Flood Fragility Analysis for Bridges with Multiple Failure Modes. Advances in Mechanical Engineering, Vol. 9, No. 3, pp. 1-11, 2017.

[12] Y.-J. Lee, J. Song, and E. J. Tuegel. Finite Element System Reliability Analysis of a Wing Torque Box. Proc. 10th AIAA Nondeterministic Approaches Conference, April 7–10, Schaumburg, IL., USA, 2008.