Transactions of the Korean Nuclear Society Autumn Meeting
Changwon, Korea, October 24-25, 2024

A study on finite element analysis of seismic response considering boundary nonlinearity of surface to surface contact

Dong-Hyun Kim [a*], Hyun-Jung Kim [a], Young-Wook Tahk [a]
[a]*Multi-purpose Small Reactor Fuel Development Division, Korea Atomic Energy Research Institute, 111 Daedeok-daero 989 beon-gil, Yuseong-gu, Daejeon 34057*
[*]*Corresponding author: donghyunkim@kaeri.re.kr*

Keywords : seismic analysis, finite element method, nonlinear boundary condition, surface to surface contact

1. Introduction

Seismic analysis is crucial in the nuclear industry to ensure the safety and reliability of facilities such as power plant and research reactor. Earthquakes can cause significant damage to these facilities, potentially leading to radioactive leaks that pose severe environmental and public health risk. The seismic analysis begins with site-specific investigations, including geological surveys and historical seismic data analysis. Structural modeling is then performed to create a comprehensive 3D representation of the facilities, and dynamic simulations, such as time history and response spectrum, are conducted using finite element models for the seismic analysis [1].

Many seismic analyses using the finite element method are known to be performed without considering nonlinearities, including material, geometrical, and boundary ones [2,3]. The primary reason for this is complexity and computational intensity of accurately modeling nonlinear behavior, which can be time-consuming and resource-intensive. However, some components in the facilities, such as relatively lightweight ones, may require nonlinear analysis due to several reasons. For example, during severe loading conditions, the components which experience large deformation or changes in boundary conditions can lead to nonlinear interactions between components and their supports or foundation.

Therefore, in this study, the finite element analysis of the seismic response considering the boundary nonlinearity was conducted using the surface to surface contact elements.

2. Analysis model and method

This section describes the finite element model and the method employed in the seismic analysis. For the analysis, commercial software ANSYS was utilized [4].

Fig. 1 shows a specific structure constructed using finite element of SOLID185. The structure is composed of one holder and eight plates. Eight plates are inserted within the slot machined inside the holder. There are gaps between the plate and the slot, which forms a nonlinear boundary condition in seismic analysis. The plates are constrained in the negative Y-direction (longitudinal) while allowing free boundary conditions in the remaining directions.

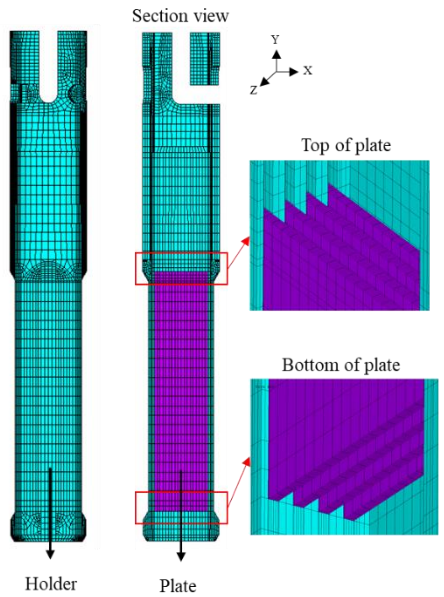

Fig. 1. Finite element model for the seismic analysis

Fig. 2 shows contact elements between plate and holder for the nonlinear boundary condition. Between the plate and holder, an initial gap exists in the X (width) and Z (thickness) directions, 0.5 mm and 0.15 mm respectively. The contact surfaces between the holder and plate are modeled using CONTA173 and TARGE170 elements. For the contact algorithm, an augmented Lagrangian algorithm was employed. A friction coefficient of 0.3 was used. Fig. 3 shows boundary conditions. Fixed boundary conditions of holder bottom are imposed.

In general, for the effectiveness of seismic analysis, the analysis is performed by linearizing the finite model by approximating nonlinear elements such as gaps. To investigate the effect of boundary nonlinearity, only one of the plates was applied with contact elements as shown in Fig. 2, The bottom of the remaining seven plates were fully bonded to the holder, allowing sliding only in the Y-direction relative to the holder. To model the condition of downward coolant flow, an initial water pressure was imposed on the upper surfaces of the holder and plates. Table I shows material properties of the finite element model. The total weight of the holder and plate is approximately 1.5 kg, which is considered lightweight.

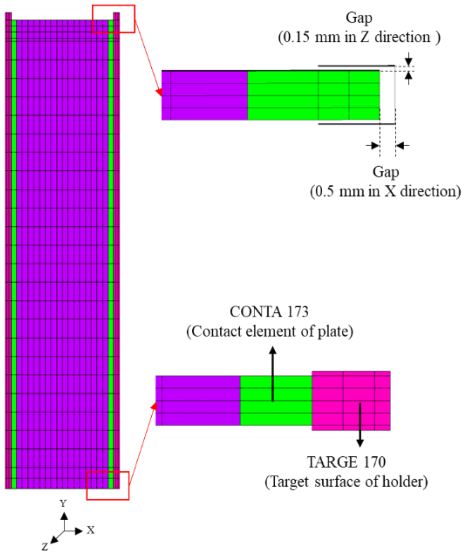

Fig. 2. Contact elements between plate and holder for nonlinear boundary condition

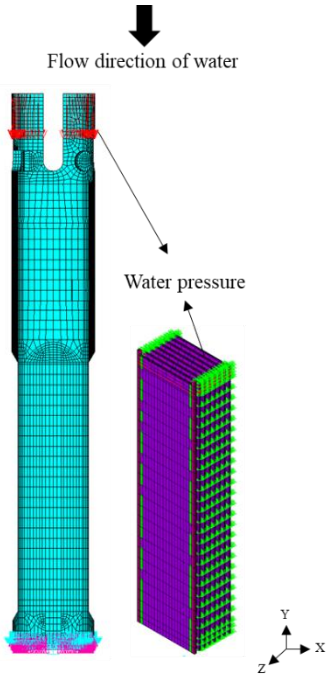

Fig. 3. Boundary conditions of the finite element model

Table I: Material properties of finite element model

	Plate	Holder
Density (kg/m^3)	2688.9	2697.8
Elastic modulus (GPa)	68.6	68.1
Poisson's ratio	0.33	0.33
Mass (kg)	0.05	1.1

For the seismic analysis, time history simulation was performed using an artificial time-acceleration history input as the seismic loading as shown in Fig. 4. Each acceleration was applied simultaneously to the model while a gravitational load of 1g was applied. During the simulation, a guide value of 3% damping was used [5].

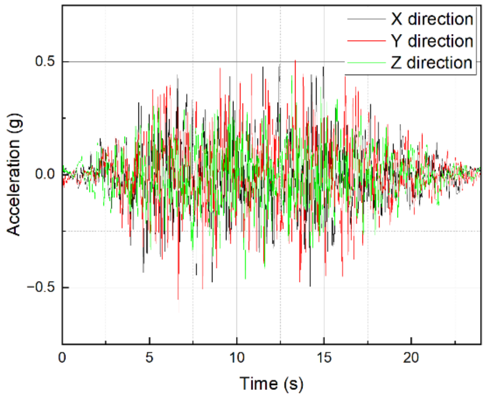

Fig. 4. Time-acceleration history for the seismic loading

3. Results and discussion

The seismic simulation was successfully completed over the 24 second seismic loading. Fig. 5 shows displacement results of the plate with the contact elements. The displacement values were extracted from the center of the top surface of the plate. The displacement in the Z-direction was observed to be the largest among the three directions. The Z-direction displacement was constrained by the 0.15 mm gap between the holder and plate, resulting in a displacement that was closed to 0.15 mm. The X-direction displacement was observed to be maximum 0.05 mm, which is significantly smaller than the 0.5 mm gap. In the Y-direction, the displacement was almost negligible due to the constraint imposed by the water pressure.

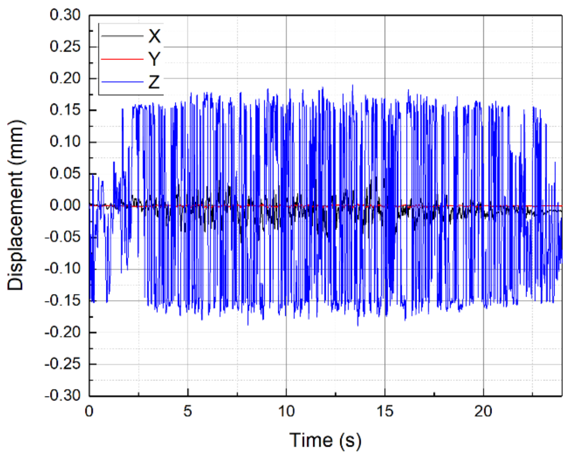

Fig. 5. Displacement results of the plate the with contact elements

Fig. 6 shows contact force results between the holder and plate. As shown in the displacement results in Fig. 5, the contact force in the Y-direction are likely due to the initial water pressure rather than the seismic loading. Most of the contact occurred in the Z-direction, which led to varying contact forces in the Z-direction over time. It is expected that the small contact force in the X-direction is attributed to frictional force.

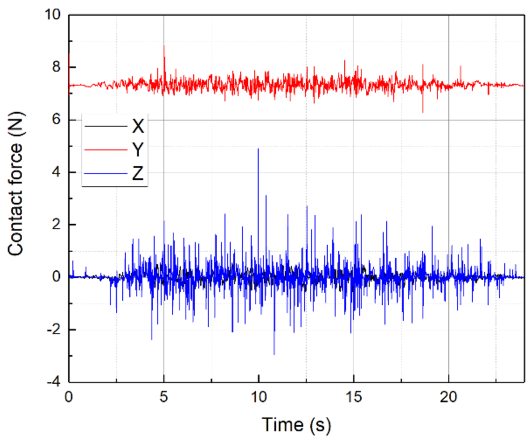

Fig. 6 shows contact force results between the holder and plate.

Fig. 7 shows stress intensity results of the plate without and with the contact element. The stress values were extracted as the maximum values among the elements at the analysis time. When considering the contact elements, it was observed that higher stress values were calculated due to the contact forces compared to linear boundary condition case where contact elements were not considered. It can be confirmed that considering contact forces yields more realistic and conservative results. Both stress results were below 4 MPa, which is significantly lower than the typical yield strength of 240 MPa of aluminum [6]. However, from the perspective of analysis time, considering contact required approximately 14 times longer analysis time compared to linear boundary condition with no contact. Therefore, for the effective seismic analysis considering the contact, it is anticipated that the complexity of the analysis model, contact characteristics, and analysis time should be carefully considered from multiple perspectives.

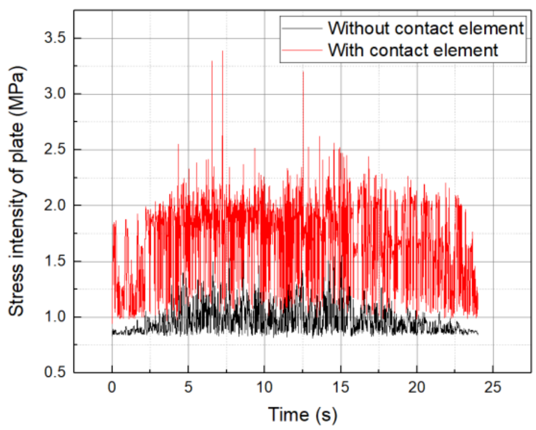

Fig. 7 Stress intensity results of the plate without and with the contact element.

4. Conclusion

In this study, a study on seismic analysis considering contact nonlinearity was conducted. For the analysis, the finite element model was developed with the contact elements. Time-history analysis was performed using the acceleration loading. The seismic analysis reflecting the contact behaviors was successfully conducted. The contact forces resulted in a more realistic and conservative stress analysis results. For the future work, additional analyses will be performed by applying various loading conditions and contact characteristics.

ACKNOWLEDGEMENT

This work was supported by the National Research Foundation of Korea (NRF) grant funded by the Korean government (MSIT: Ministry of Science and ICT) (NRF-2020M2C1A1061031).

REFERENCES

[1] R. Lo Frano, G. Pugliese, G. Forasassi, Preliminary seismic analysis of an innovate near term reactor: Methodology and application, Nuclear Engineering and Design, Vol. 240, p.1671-1678, 2010
[2] J. B. Park, S. J. Lee, E. H. Lee, N. C. Park, Y. B. Km, Seismic response of nuclear reactor vessel internals considering coolant flow under operating conditions, Nuclear Engineering and Technology, Vol. 51, p. 1658-1668, 2019
[3] L.L Tong, R. Duan, X. W. Cao, Seismic analysis of RCS with finite element model for advanced PWR, Progress in Nuclear Energy, Vol. 79, p. 142-199, 2015
[4] ANSYS release 19.0 Structural Analysis guide, ANSYS INC., Southpointe, 2600 ANSYS Drive Canonsburg, PA 15317, USA
[5] Damping values for seismic design of nuclear power plants, Regulatory guide 1.61, Rev. 1, USNRC, 2007.
[6] American Society of Mechanical Engineers (ASME), ASME Boiler and Pressure Vessel Code: Section 2 - Materials. Part D: properties. ASME, 2004