Impact Analysis for Fuel Assemblies in Spent Fuel Storage Rack

J. H. Oh*, J. S. Ryu
Korea Atomic Energy Research Institute,
989-111 Daedeok-daero, Yuseong-gu, Daejeon,
305-353, Korea

*Corresponding author: jinhooh@kaeri.re.kr

ABSTRACT

The design and structural integrity evaluation of a spent fuel storage rack (SFSR) utilized for storing and protecting the spent fuel assemblies generated during the operation of a reactor are very important in terms of nuclear safety and waste management. The objective of this study is to show the validity of the SFSR design as well as fuel assembly through a structural integrity evaluation based on a numerical analysis. In particular, a dynamic time history analysis considering the gaps between the fuel assemblies and the walls of the storage cell pipes in the SFSR was performed to check the structural integrity of the fuel assembly and storage cell pipe.

1. Introduction

After the accident of the Fukushima Daiichi Nuclear Power Plant, concerns regarding safety, security, and/or the environment of the nuclear industry have greatly increased. In particular, a number of researchers have given special attention to a spent fuel storage and handling system because a hydrogen explosion caused by the melting of zircaloy surrounding the spent fuel assembly was experienced [1].

Generally, there are two kinds of pools, a reactor pool and a service pool, in a research reactor facilities. The research reactor and related equipment such as beam port assembly are installed in the reactor pool. The service pool is mainly utilized for storing and protecting the spent fuel assemblies. Specifically, the spent fuel assemblies taken out of the reactor core are transported and stored into the SFSR in the service pool using a proper irradiated object handling system such as a crane-type facility. The service pool is connected to the reactor pool. De-mineralized water is filled in the service pool to remove decay heat and shield from radiation generated from the spent fuel assemblies. In this study, the structural design of SFSR and integrity evaluation of the fuel assemblies in an SFSR are focused upon. Most of the structure used to store spent fuel assemblies has a simplified single structure. However, the designed storage structure in this study consists of double structures composed of racks and a support frame [2]. Thus, when a seismic event occurs, since the vibration is transferred to the spent fuel assemblies through the support frame and SFSR, the structural stability is ensured. For evaluating the structural integrity of the spent fuel assembly, the modal analysis and linear/nonlinear time history analyses have been performed. In particular, the nonlinear time history analysis using the implicit code has been performed for predicting the contact sliding quantity of the support frame. Finally, the time history data of the SFSR obtained from the analysis is used as the input data of the dynamic time transient analysis for evaluating the impact between the fuel assembly and storage cell pipe. These structural evaluation results contribute to an improvement of the safety of fuel storage.

2. Structural Analysis

For an evaluation of the structural integrity of the fuel assembly and storage cell pipe in the SFSR, a modal analysis and dynamic time history analysis have been performed. To investigate the dynamic characteristics of the fuel assembly and storage cell pipe, a modal analysis was performed. The typical measures of the dynamic characteristics such as natural frequencies and mode shapes were obtained. The objective of the dynamic time history analysis is to check whether the impact between the fuel assembly and storage cell pipe occurs or not under the seismic event. If the impact happened, the impact analysis is carried out for the evaluation of structural integrity. For implementing the analyses, 3-D finite element models of an SFSR, support frame and fuel assembly have been developed.

2.1 Finite Element Modeling

The finite element model of the SFSR, their support frame and fuel assembly are developed by using ANSYS v.14.5 software. Fig. 1 shows the configuration of the reactor and service pool. Fig. 2 represents the configuration of the SFSR and their support frame. Figs. 3 and 4 show the finite element models for structural analyses of the SFSR, support frame, storage cell pipe and fuel assembly. The elements used in the analysis model are the solid element, the shell element, the nonstructural mass element, the beam element and the rigid element. The total number of elements and nodes of the SFSR are 214,243 and 69,667, respectively. The total number of elements and nodes of the support frame are 14,256 and 15,264, respectively. The total number of elements and nodes of the fuel assembly are 293 and 147, respectively. The storage cell pipe is modeled using solid elements. The top and bottom parts of the storage cell are connected to the guide and base plates. The SFSRs and support frame are made of stainless steel to prevent corrosion. The fuel assembly is made of aluminum, U_3Si_2, and AG3NE. The fuel assembly is modeled using the beam elements because the solid and shell elements require lots of computation cost. The fuel assembly is composed of 15 sections, and each section has cross-section information such as the moment of inertia, shear center, and area. The material properties are shown in Table 1. The main components affected by the hydrodynamic mass are the storage cell pipe/base plate of the SFSR, the bottom/middle frame of the support frame, and the fuel assembly. The hydrodynamic masses of the SFSR are about 550 kg in the horizontal direction and 760 kg in the vertical direction. Those of the support frame are about 1530 kg, 1720 kg, and 3340 kg in the horizontal and vertical direction, respectively. Those of the fuel assembly are about 6.7 kg in the horizontal direction.

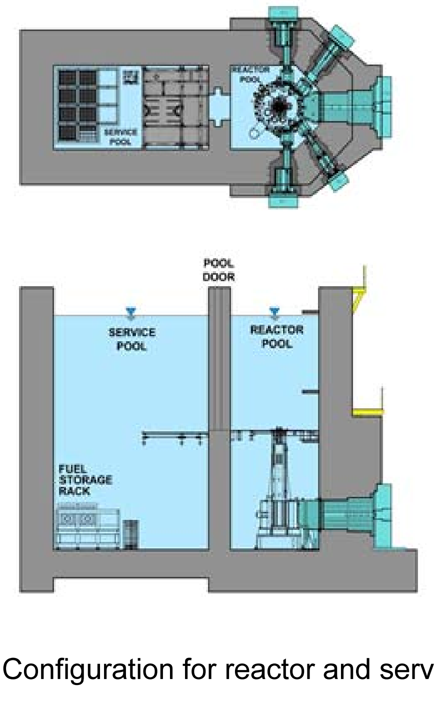


Fig 1. Configuration for reactor and service pool

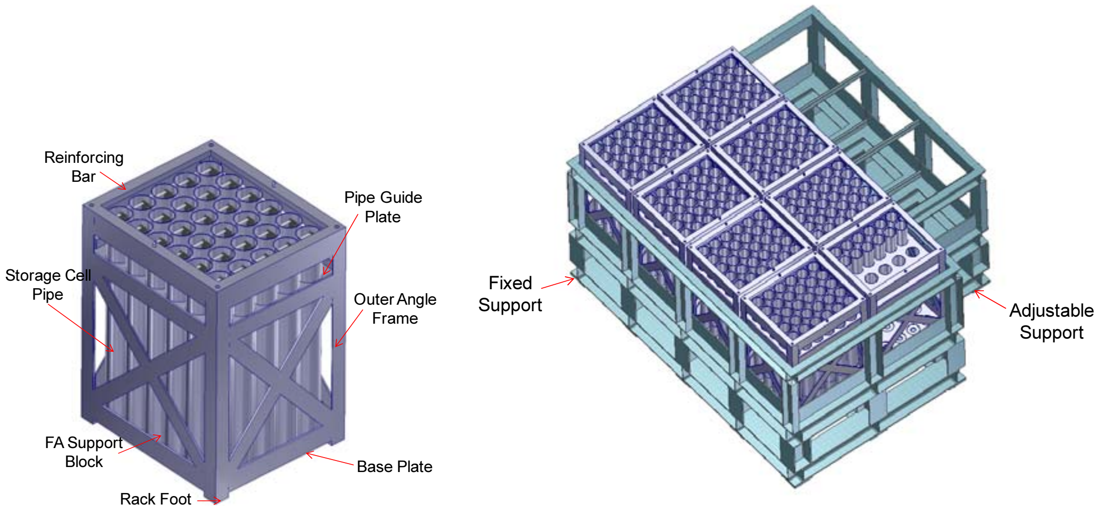


Fig 2. Configuration for SFSR and support frame

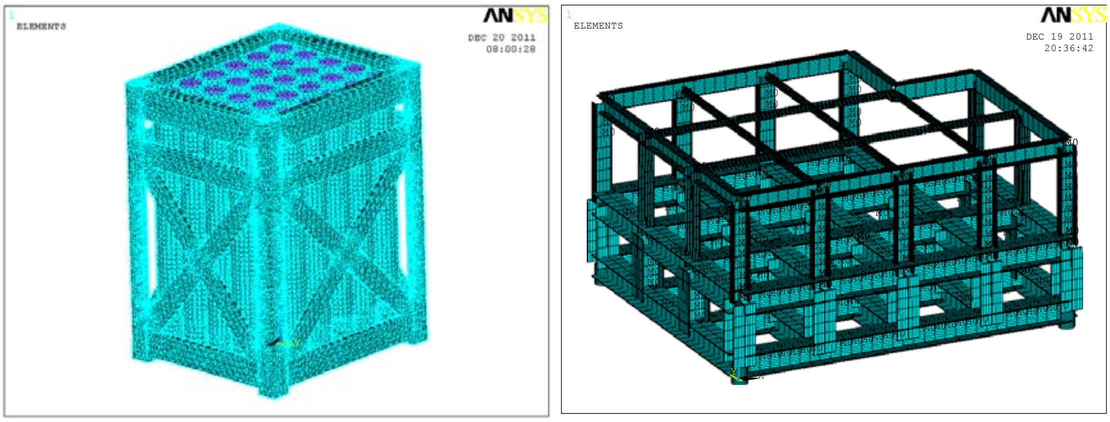


Fig 3. Finite element model for SFSR and support frame

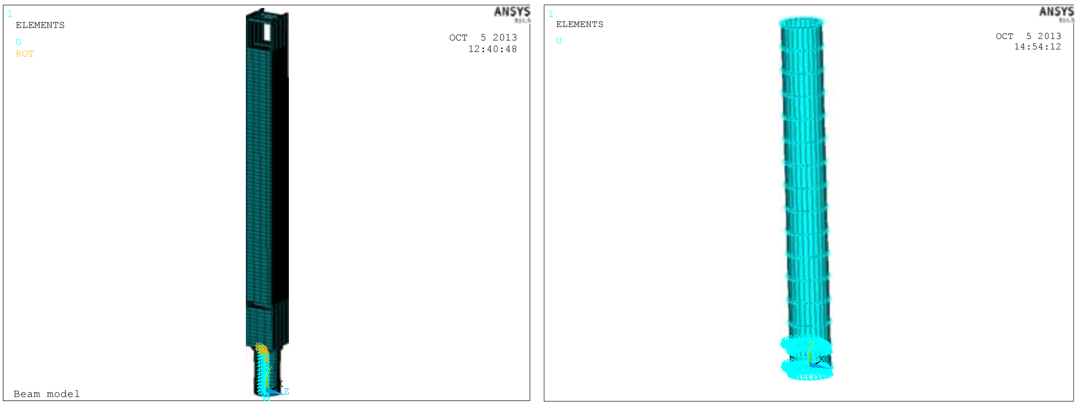


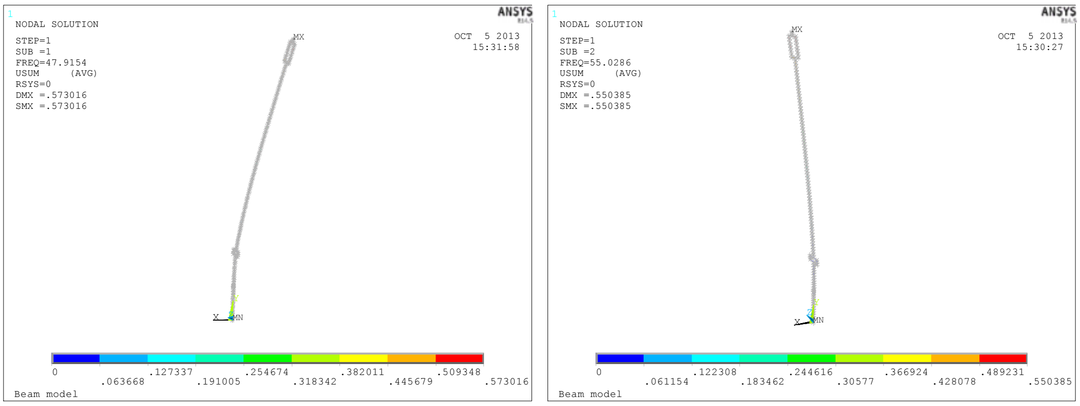
Fig 4. Finite element model for fuel assembly and storage cell pipe

Properties	Stainless Steel	Al 6061-T6	AG3NE
Young’s modulus	1.98E+11 N/m^2	7.04E+10 N/m^2	6.93E+10 N/m^2
Poisson’s ratio	0.29	0.33	0.33
Density	7930 kg/m^3	2700 kg/m^3	2700 kg/m^3

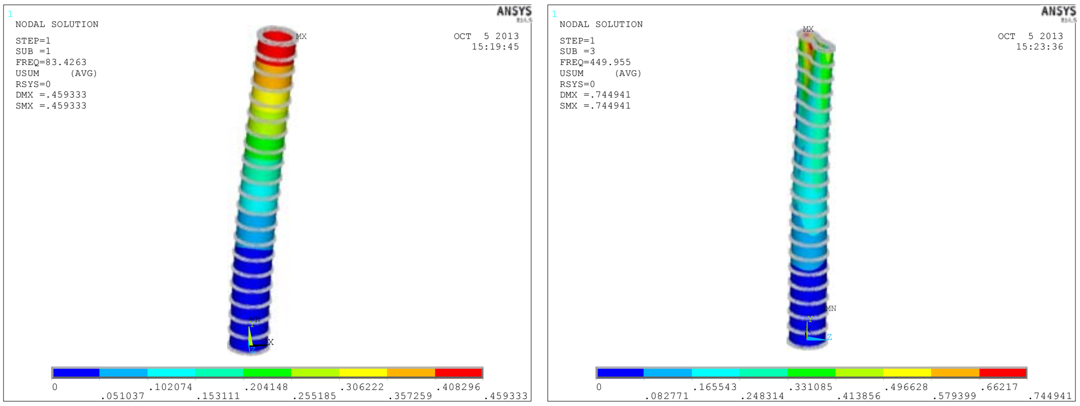
Tab 1: Material properties of stainless steel, aluminum and AG3NE

2.1 Modal Analysis

A modal analysis was carried out first to check the dynamic characteristics of the fuel assembly and storage cell pipe in the SFSR. The hydrodynamic effect caused by the fluid is considered using the added mass method. The added mass of the fuel assembly was calculated through the correlation with the results of 3-D fluid-structure interaction analysis. Fig. 5 shows the mode shape of the fuel assembly and storage cell pipe. The first bending mode shapes are similar and move in phase with each other. The boundary conditions were properly imposed considering the condition when the fuel assembly is placed in the storage cell pipe. Actually, since the dowel pin and support block prohibit the fuel assembly from translating and rotating, all degrees of freedom at end fitting part, a lower part of fuel assembly, were constrained. The frequencies of the fuel assembly and storage cell pipe are shown in Table 2. The frequencies of the first modes are 47.9 Hz and 83.4 Hz, respectively. Since the frequencies are much higher than 33 Hz, they are regarded as the rigid structure. Therefore, it is confirmed that there is no need to do resonance avoidance design such as a response spectrum analysis to verify the structural integrity of the fuel assembly.



(a) First mode shape of fuel assembly (b) Second mode shape of fuel assembly



(a) First mode shape of storage cell pipe (b) Second mode shape of storage cell pipe

Fig 5. First and second mode shape of fuel assembly and storage cell pipe

Frequencies	Fuel Assembly	Storage Cell Pipe
First Freq.	47.9	83.4
Second Freq.	55.0	437.2
Third Freq.	317.4	470.8
Fourth Freq.	321.5	486.8

Tab 2: Frequencies of storage cell pipe and fuel assembly

2.2 Dynamic Time Transient Analysis

The fuel assemblies will be able to rattle inside their storage locations during an earthquake. It is thus highly important to check the integrity of the fuel assembly under impact loading. The input data for a rattling behavior is obtained through a nonlinear time history analysis of the support frame including the SFSRs [3]. The input data are extracted at the position of the SFSR top frame because the relative displacement at that position is larger than those at the other position. The excitation position is the lower part of fuel assembly and storage cell pipe where the degree of freedom is constrained. Fig. 6 shows the calculated time history data. In the impact analysis, the important check point is how to model the structures by using finite element method. Generally, the gap between the fuel assembly and cell pipe used to be modeled using a 3-dimensional combination element to simulate the rattling phenomenon of the fuel assembly. However, an accurate representation of the impact stiffness of the combination element is difficult. Also, the seismic response may also be very sensitive to variations in impact stiffness. Thus, the node to surface contact condition is applied between the fuel assembly and storage cell pipe for obtaining a better solution.

In this study, the dynamic time history analysis is first performed to check the occurrence of impact between them because the impact analysis requires lots of computation costs. Dynamic time history results are shown in Fig. 7 and Fig. 8. The Fig. 7 demonstrates that any impact doesn't happen because the calculated maximum gap distance (0.91 mm) is less than the minimum gap (1.618 mm) between the fuel assembly and storage cell pipe. The Fig. 8 shows that the maximum stresses of them occur on the lower part of fuel assembly and pipe. Also, when the SSE (Safe Shutdown Earthquake) loading are applied to the SFSR and support frame [4], it is confirm that the maximum axial direct stress of fuel assembly is about 2.12MPa and the stress intensity of storage cell pipe is about 9.15MPa. The dynamic time history by OBE (Operating Basis Earthquake) excitation was not performed because the SSE acceleration is larger than that of the OBE. Finally, these analysis results show that maximum stresses are within the structural yield stresses and the possibility of impact between them is rare. Even if the impact between the fuel assembly and cell pipe happened, it would not substantially affect the structural integrity of them because the magnitude of the impact force seems to be very small.

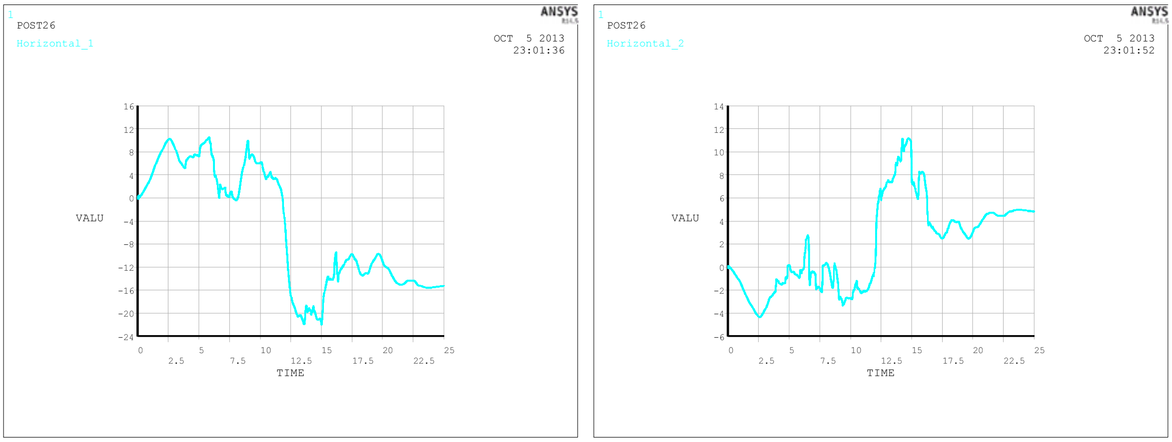
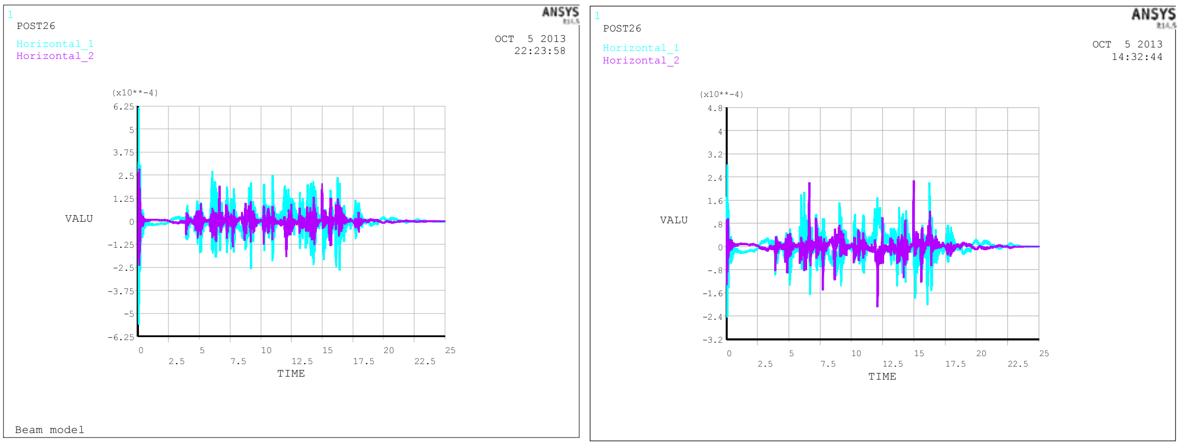
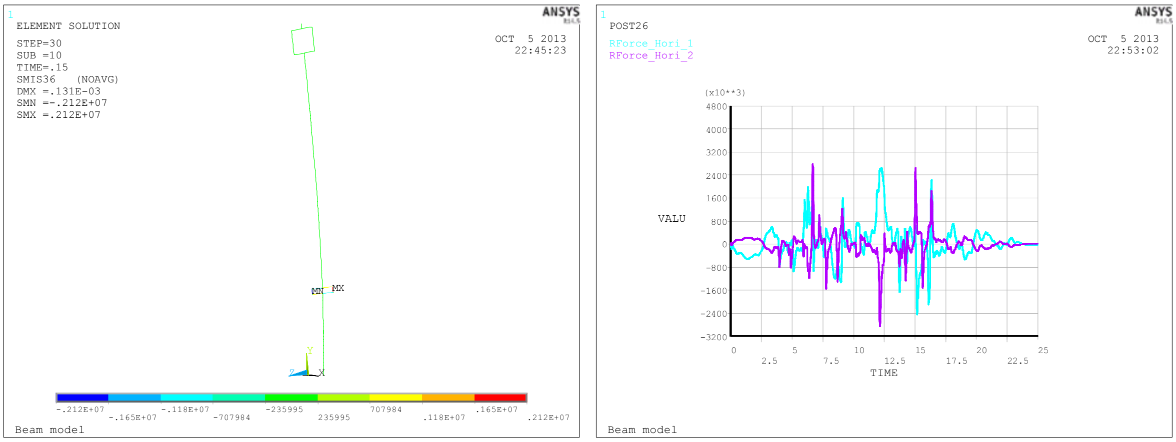


Fig 6. Time history input data for dynamic time history analysis between fuel assembly and storage cell pipe

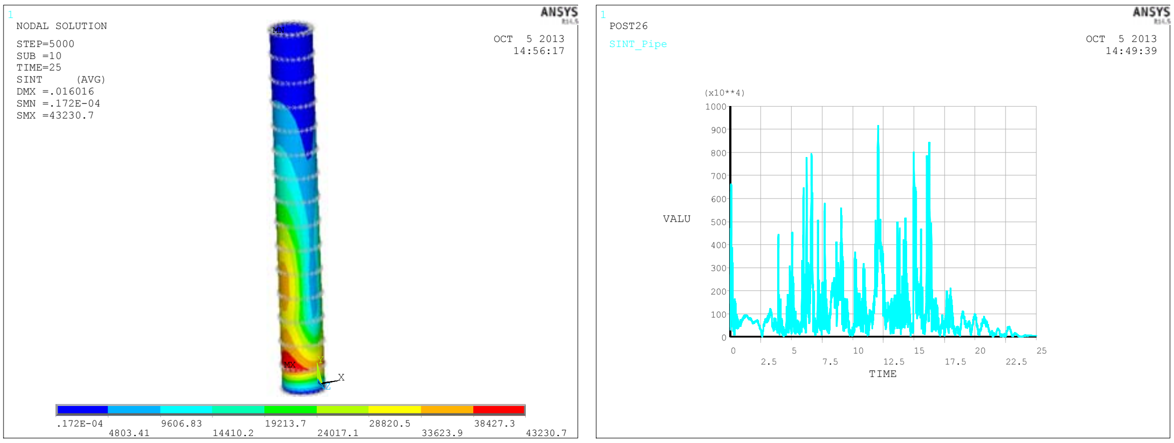


(a) Fuel assembly (b) Storage cell pipe

Fig 7. Relative displacement of top and bottom position according to the time variation



(a) Axial direct stress of fuel assembly (b) Reaction forces of fuel assembly



(c) Stress intensity of storage cell pipe (d) Stress intensity variation

Fig 8. Axial direct stress and stress intensity of the fuel assembly and storage cell pipe

3. Conclusion

The structural integrity of the fuel assembly and the storage cell pipe in the SFSR has been evaluated through a dynamic time history analysis based on the time history data. The 3-D nonlinear time history analysis was first performed to obtain the input data for rattling the fuel assembly. The fuel assembly was modeled with a beam element including the cross section information. The added mass method was adopted to consider the hydrodynamic effect. The accurate added mass was calculated by correlating the modal analysis results of the current beam model with those of the 3-D fluid-structure interaction model. In addition, the dynamic characteristic of the fuel assembly and storage cell pipe was also checked by means of a modal analysis. Consequentially, it was verified that the resonance avoidance design does not need to be performed because the first frequency of the fuel assembly is above 33 Hz, which is a yardstick of the resonance design.

Finally, it was observed through a dynamic time history analysis that there is no damage to the fuel assembly and storage cell pipe because the impact of them does not occur and the stresses calculated by the SSE seismic loads are within the yield stresses. However, since these results are only based on the numerical simulations, it is necessary to demonstrate the validity and robustness of the fuel assembly and storage cell pipe by implementing nonlinear impact analysis using the maximum velocity of fuel assembly obtained from this study.

4. References

[1] Katsumi Shozugawa, Norio Nogawa, and Motoyuki Matsuo, 2012, “Deposition of fission and activation products after the Fukushima Dai-ichi nuclear power plant accident Original Research Article,” Environmental Pollution, Volume 163, Pages 243-247.
[2] Regulatory Guide 1.13, 2007, “Spent Fuel Storage Facility Design Basis,” U.S. Nuclear Regulatory Commission.
[3] Regulatory Guide 1.61, 2007, “Damping Values for Seismic Design of Nuclear Power Plants,” U.S. Nuclear Regulatory Commission.
[4] Regulatory Guide 1.29, 2007, “Seismic Design Classification,” U.S. Nuclear Regulatory Commission.