Transactions of the Korean Nuclear Society Autumn Meeting
Changwon, Korea, October 21-22, 2021

Simulation of Fuel Assembly Mechanical Behavior

Youngik Yoo*, joongjin Kim, Namgyu Park, Sangyoun Jeon, Hyeongkoo Kim
yiyoo@knfc.co.kr

Global Leading Nuclear Energy Partner
KEPCO NF

1 Introduction

In Korea, various types of spent nuclear fuels(SNFs) are stored and soon reach saturation. Therefore, SNFs must be moved to a dry storage facility, and integrity evaluation of fuels must be performed to ensure safety.

The purpose of this study is to develop a finite element model to evaluate the mechanical integrity of a nuclear fuel assembly. A model that satisfies beginning-of-life(BOL) static and dynamic mechanical behaviors was developed and verified using the BOL test results. In the near future, the SNF model verification will be performed using the end-of-life(EOL) test results.

2 Model development

The target fuel assembly(FA) is 17×17 type. The finite element model of the FA consists of all major components including the skeleton(top and bottom nozzles, top and bottom grids, five mid grids, and 16 guide tubes) and 264 fuel rods. Beam and spring elements were used to characterize all the components that were modeled considering their nominal dimensions and geometric shapes. ABAQUS v6.14-3 was used to create the parametric finite element model. Parameters for adjusting the models were defined to match the test results. The typical 3D FA and component models is shown in Fig. 1.

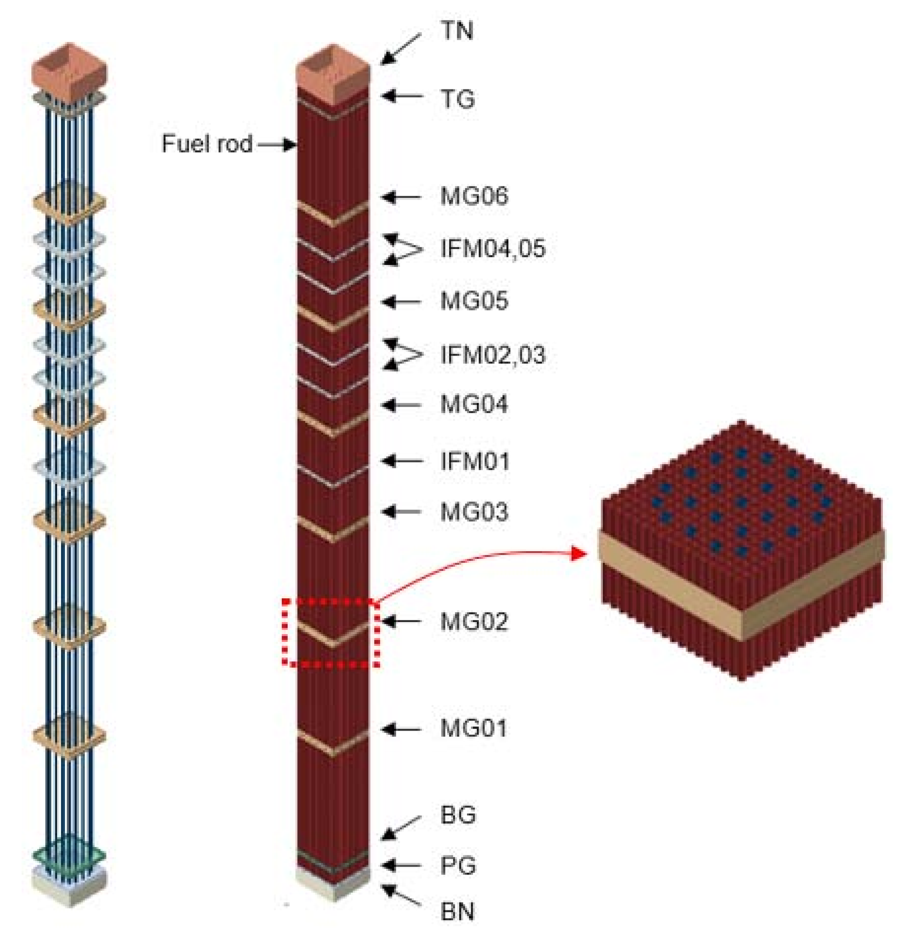

Fig. 1. Fuel assembly model

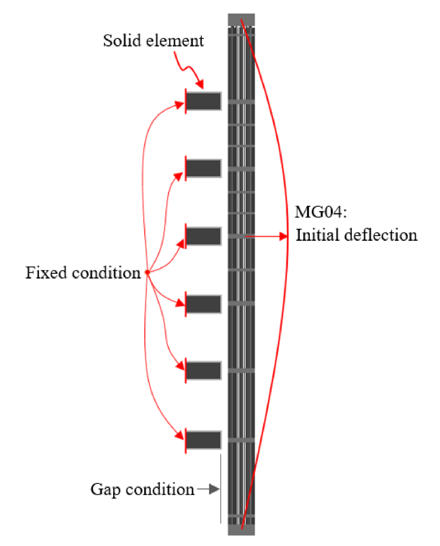

Fig. 5. Lateral Impact model

3 Model validation

3.1 Modal Analysis

Using the finite element model developed in this study, the modal analysis was performed. The boundary conditions were added at the surface, which was the contact area between top/bottom plates and the FA. The natural frequencies and mode shapes of the FA are shown in Fig. 2 and Fig. 3, which show that the normalized analysis results are in good agreement with the test results. Therefore, the FA finite element model is valid from the dynamic viewpoint.

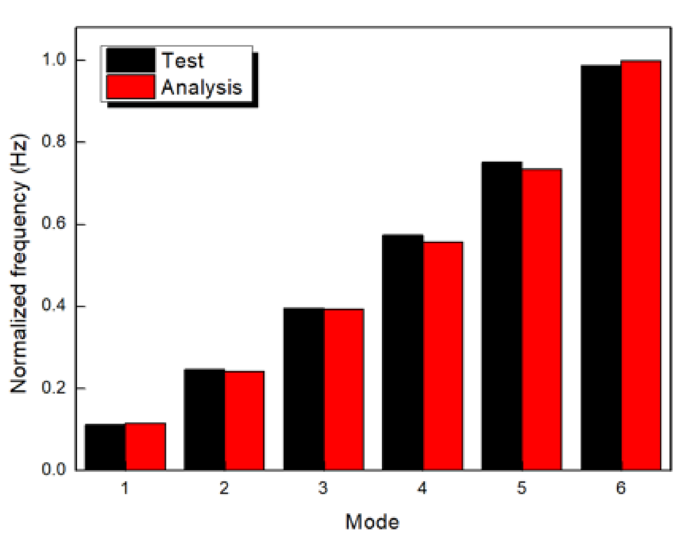

Fig. 2. Natural frequency results

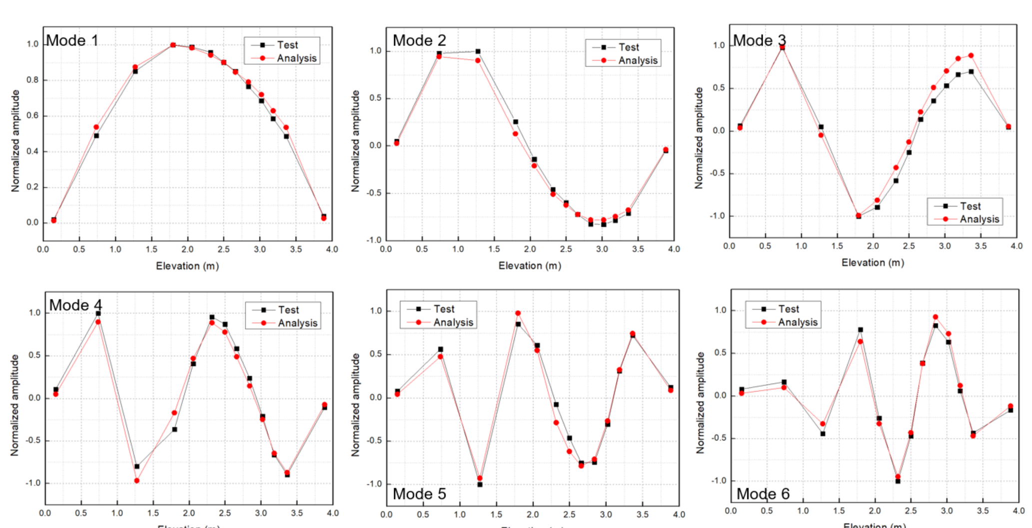

Fig. 3. Mode shape

3.2 Lateral deflection analysis

Lateral deflection of 25 mm was applied to the mid grid (MG03~05). The load-displacement analysis results(Fig. 4) were very similar to the tests, and the lateral stiffness, friction and residual deflection were well simulated. The developed model is also valid in static mechanics.

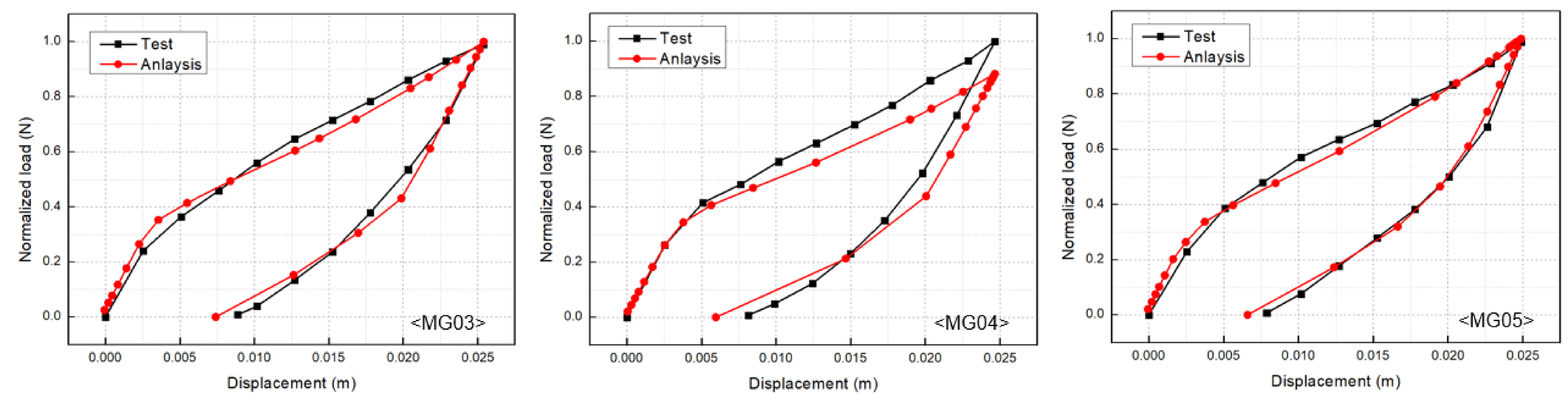

Fig. 4. Lateral deflection

3.3 Lateral impact analysis

Solid elements were added to the left of the FA model, and each solid surface was fixed(Fig. 5). The initial deflection at mid grid 04 increased from 0.2 inch to 0.1 inch in increments of 0.4 inch, and, after the initial deflection condition was removed, reaction forces at solid element were measured.

Fig. 6 shows the lateral impact analysis results. As shown in the figure, the analysis results match well to the test data. Therefore, the analysis model is valid for the lateral impact simulation.

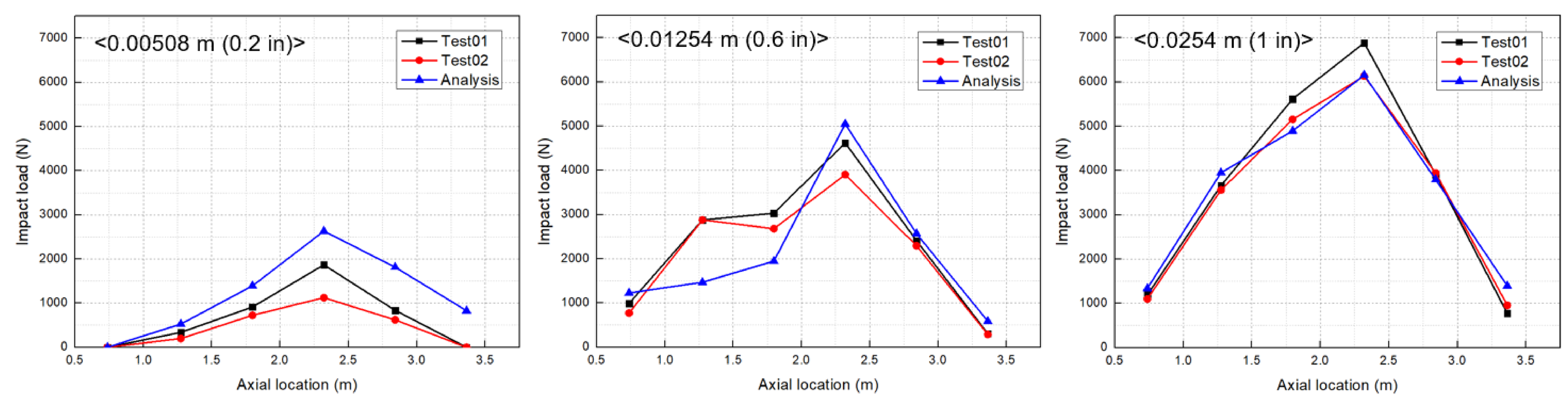

Fig. 6. Lateral Impact load

3.4 Axial impact analysis

Solid element was added to the bottom of the FA model, and bottom surface was fixed(Fig. 7). The initial drop height increased from 0.2 inch to 1.0 inch in increments of 0.2 inch. Fig. 8 shows the lateral impact analysis result, which agrees well with the test.

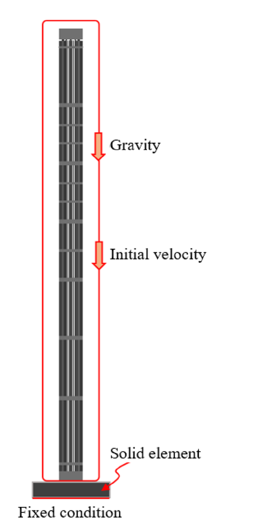

Fig. 7. Axial Impact model

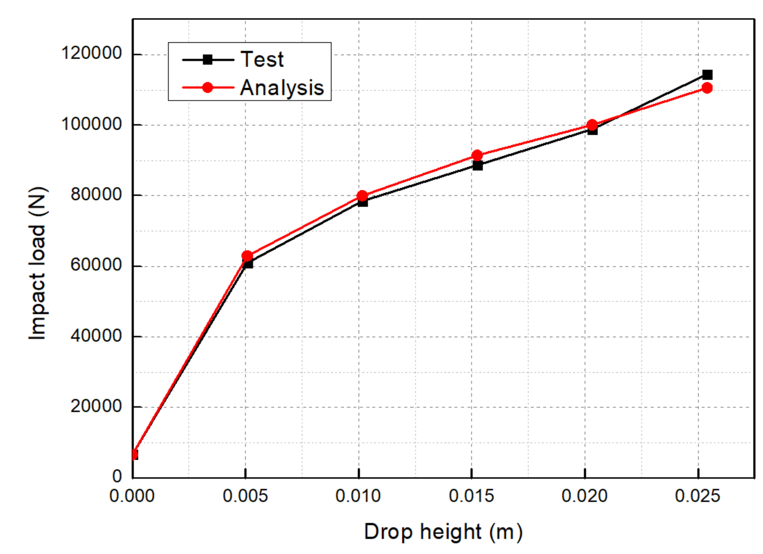

Fig. 8. Axial Impact load

4 Conclusion

In this study, a 3D finite element model is developed. The model was verified by comparison with the BOL test data, and the static and dynamic behaviors were well simulated. In the near future, the SNF model verification will be performed using the end-of-life(EOL) test results. And, this model will be used to evaluate the fuel damage under normal and accident conditions of SNF.

KEPCO NF