20th International Conference on Structural Mechanics in Reactor Technology (SMiRT 20)
Espoo, Finland, August 9-14, 2009
SMiRT 20-Division 6, Paper 1858

Development of the Simplified Fuel Assembly Model for the Fuel Assembly SSE and LOCA Analysis

Kyou Seok Lee[1], Sang Youn Jeon[1], and Hyeong Koo Kim[1]

[1]Korea Nuclear Fuel (493, Deogjin-Dong, Youseong-Gu, Daejeon, 305-353, Korea
Tel : +82-42-868-1185, Fax : +82-42-868-1149, Email : kslee@knfc.co.kr

Keywords: Simplified Fuel Assembly Model, SSE and LOCA Analysis, Optimization

1. ABSTRACT

Under the Safe Shutdown Earthquake (SSE) and Loss of Coolant Accident (LOCA) events, the fuel assembly lateral deflection and grid impact force between fuel assemblies are obtained by the dynamic transient analysis for the reactor core finite element model.

Since a reactor core consists of a large number of fuel assemblies which have non-linear dynamic characteristics and complicate structural geometry, the dynamic behavior of the core under the postulated events is very difficult to analyze. Therefore, it is necessary that fuel assembly model is simplified with considering the dynamic non-linear characteristics in core analysis.

In this study, a simplified fuel assembly finite element model has been developed using the optimization technique of ANSYS code and the configuration of the model was proposed through by sensitivity study. In order to verify the model, the simulations for static test, pluck vibration test, pluck impact test were performed. The results show a good correlation with fuel assembly mechanical test results.

2. INTRODUCTION

The load on the fuel assembly under accidents and postulated events like Safe Shutdown Earthquake (SSE) and Loss of Coolant Accident (LOCA) events shall not results in permanent deformation that would prevent effective emergency cooling of the fuel or that would prevent safe reactor shutdown. Under the SSE and LOCA events, the fuel assembly lateral deflection and grid impact force between fuel assemblies are obtained by the dynamic transient analysis for the reactor core finite element model.

The impact behavior between fuel assemblies shows non-linear characteristics, because fuel assembly shows non-linear dynamic characteristics and its structural geometry is complicated. Furthermore, since a reactor core consists of a large number of fuel assemblies, the dynamic behavior of the core under the postulated events is very difficult to analyze due to the nonlinearity and huge model

size. Therefore, it is necessary that fuel assembly model is simplified with considering the dynamic non-linear characteristics in core analysis. However, the simplification of the fuel assembly model is very difficult, because fuel assembly has very complicated geometry and many non-linearities. Until now many researches have been performed to simplify the fuel assembly.

In this study, a simplified fuel assembly finite element model for the Westinghouse type 17×17 RFA has been developed using the optimization technique. To obtain the simplified model, the optimization algorithm of ANSYS code was used, and the configuration of the model was determined by the sensitivity study. The simulations for static test, pluck vibration test, pluck impact test were performed using the model. The simplified fuel assembly model was verified by comparison with fuel assembly mechanical test results.

3. OPTIMIZATION DESIGN TECHNIQUE

3.1 General description of optimization design

The independent variables in an optimization analysis are the design variables. The vector of design variables is indicated by:

$$x = [x_1, x_2, x_3, \ldots, x_n] \tag{1}$$

Design variables are subject to n constraints with upper($\bar{x}_i$) and lower limits($\underline{x}_i$), that is,

$$\underline{x}_i \le x_i \le \bar{x}_i \quad (n=1, 2, 3, \ldots, n) \tag{2}$$

where, n = number of design variables.

The design variable's constraints are often referred to as side constraints and define what is commonly called feasible design space.

Now, minimize

$$f = f(x) \tag{3}$$

Subject to

$$g_i(x) \le \bar{g}_i \qquad (i=1, 2, 3, \ldots, m_1) \tag{4}$$

$$h_i(x) \le \bar{h}_i(x) \qquad (i=1, 2, 3, \ldots, m_2) \tag{5}$$

$$\underline{w}_i \le w_i(x) \le \bar{w}_i \qquad (i=1, 2, 3, \ldots, m_3) \tag{6}$$

where:

f : object function

g_i, h_i, w_i = state variables containing the design, with underbar and overbars representing lower and upper bounds respectively

$m_1+m_2+m_3$ = number of state variables constraints with upper and lower limit values

The subproblem approximation method of ANSYS was used to optimize the simplified fuel assembly model. This optimization method can be described as an advanced, zero-order method in that requires only the values of the dependent variables (object function and state variables) and not their derivatives. This optimization method is summarized as follow:

1) The objective function approximation is used to locate the minimum
2) State variables approximations are used to constraint the design
3) After the approximation are used to locate the minimum object function in design space, the actual object function and state variables are evaluated at that location
4) A least squares fit of all available design sets is used to form the approximations:

$$H = a_0 + \sum_{n=1}^{N} a_n X_n + \sum_{n=1}^{N} a_n X_n^2 + \sum_{m=1}^{N-1} \sum_{n=m+1}^{N} c_{mn} X_m X_n \qquad (7)$$

where:

H : approximation of object function or state variables

X_n: design variable n

N: total number of design variables

5) Subproblem approximation iterations continue until either convergence is achieved or termination occurs. Convergence is assumed when either the present design set, x^j, or the previous design set, $x^{(j-1)}$, or the best design set, $x^{(b)}$, is feasible.
6) A least squares fit of all available design sets is used to form the approximations:

$$H = a_0 + \sum_{n=1}^{N} a_n X_n + \sum_{n=1}^{N} a_n X_n^2 + \sum_{m=1}^{N-1} \sum_{n=m+1}^{N} c_{mn} X_m X_n \qquad (7)$$

3.2 Definition of optimization problem for simplified fuel assembly model

The object function of optimization problem for the simplified fuel assembly model was defined as the minimization for the difference between the test natural frequency of fuel assembly and optimized natural frequency of the model. The constraints of the problem were defined as tolerance range for frequency at the each mode.

The optimization problem for simplified model of fuel assembly was defined as follow;

- Object function: Minimize $|\ _T - \ _C|$

where:

$_T$: Target Frequency (test results)

$_C$: Converged Frequency

- Subject to $f_i \leq \pm \ _i$

where:

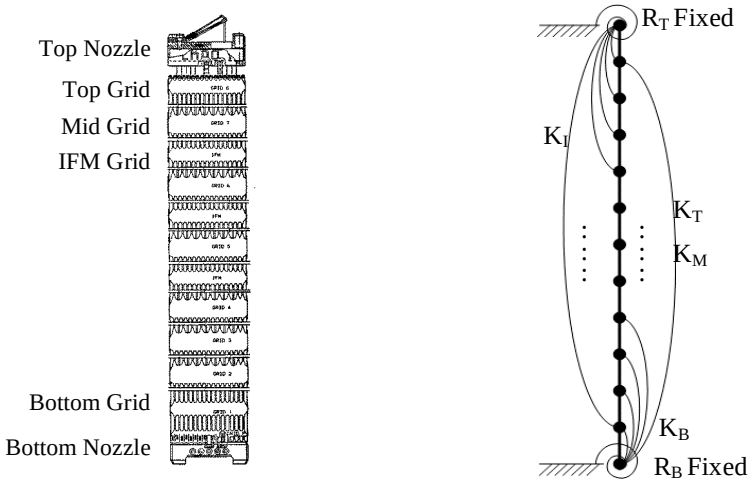
$f \ _i$: difference of each frequency at the i^{th} mode

$_i$: Converged tolerance

4. CONFIGURATION OF SIMPLIFIED FUEL ASSEMBLY MODEL

The 17×17 RFA is consisted of top and bottom nozzle, 24 guide thimbles, 1 instrumentation tube, inconel top and bottom grid, protective grid, 6 mid grids, 3 IFM grids, and 264 fuel rods as shown in Figure 1. The fuel assembly attached IFM grids cannot be simulated to be simple uniform beam structure because of the non-symmetric of IFM attached in the upper region of fuel assembly and non-linear gap characteristics between IFM dimple and fuel rod. Therefore, additional finite element to simulate these non-symmetric and non-linear properties is needed.

In this paper, to obtain the equivalent finite element model with the dynamic characteristics of fuel assembly, the design variables of model were considered to be uniform beam rigidity (I) for fuel rods and guide thimbles, the rotational rigidities (R_T and R_B) at top and bottom end to simulate the interface rigidities between top/bottom nozzles and core plates, the rotational springs (K_T: top grid, K_B: bottom grid, K_M: mid grid, K_I:IFM grid) between each grid and top and bottom ends to simulate the friction resistance between grid spring and fuel rod. The configuration of the model is shown in Figure 1.



(a) 17×17 RFA (b) Simplified Model

Fig. 1 Fuel Assembly and Simplified Model Configuration

The sub-problem approximation method in ANSYS was used for the optimization of model, and the design variables of beam rigidity and the stiffness for each rotational spring were optimized.

The preliminary optimization was performed to verify the configuration of simplified fuel assembly model. After preliminary optimization, the rotational rigidities (R_T and R_B) at top and bottom end to simulate the interface rigidities between top/bottom nozzles and core plates were converged to be very high value. Therefore, the fixed boundary conditions at top and bottom end of the model were imposed as shown in Figure 2. That is, the design variables for R_T and R_B were removed in the configuration of the model.

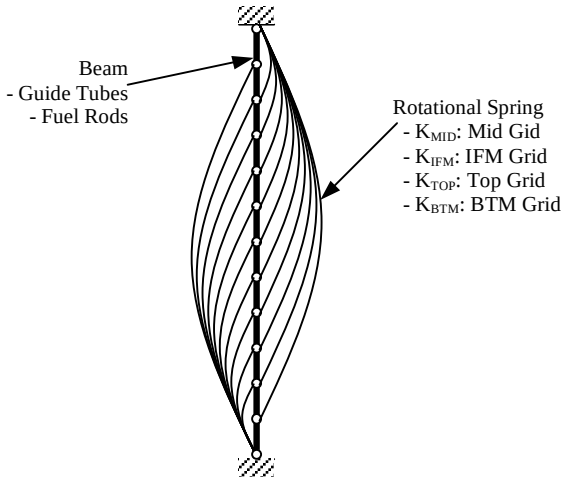


Fig. 2 Simplified Fuel Assembly Model

5. ANALYSIS AND RESULTS

In order to obtain the simplified fuel assembly model which has the equivalent dynamic characteristics with real fuel assembly, an optimization analysis for the model which has initial design variables was performed. The optimized natural frequencies and mode shapes of the model was obtained as the finial results of optimization analysis. The simulations for static test, pluck vibration test, and pluck impact test were performed to verify the optimized fuel assembly model.

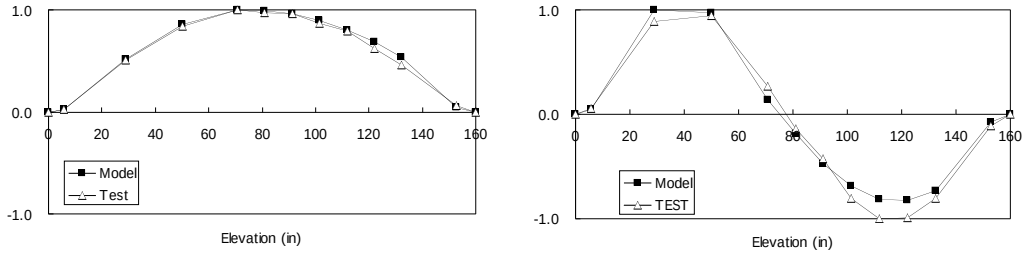
5.1 Optimization and Modal Analysis

The optimization for the modified model, which has the design variables of uniform beam and rotational spring for each grid, was performed as same method as preliminary optimization. The initial values of beam rigidity (I) was the sum of the beam rigidities of guide thimbles and fuel rod. The initial values of grid rotational spring stiffness were 5.0E5 lbs/in for mid grid, top grid and bottom grid, and 2.5E5 lbs/in for IFM grid. The convergence rates were 1.0% for the 1^{st} mode, 1.5% for the 2^{nd} mode, 2.0% for the 3^{rd} mode, 2.5% for the 4^{th} mode, and 5.0% for the 5^{th} and 6^{th} mode.

The optimized model for modal analysis has a good correlation with fuel assembly mechanical test results as shown in Table 1 and Figure 3.

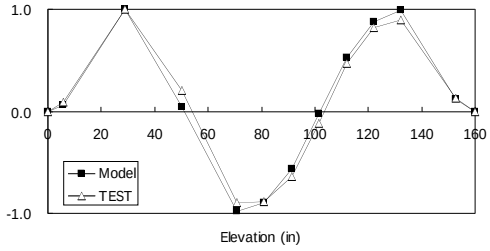
Table 1 Comparison of Natural Frequencies with Test Results

Mode		1st	2nd	3rd	4th	5th	6th
Frequency (Hz)	Test	3.65	7.8	12.3	17.7	24.2	30.5
	Model	3.65	7.89	12.53	18.44	24.38	34.12

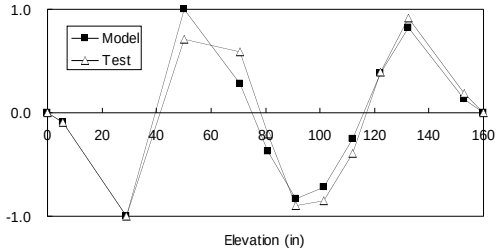


a) 1st Mode

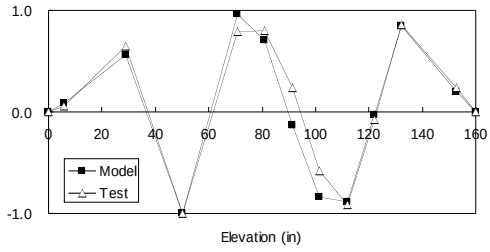
b) 2nd Mode



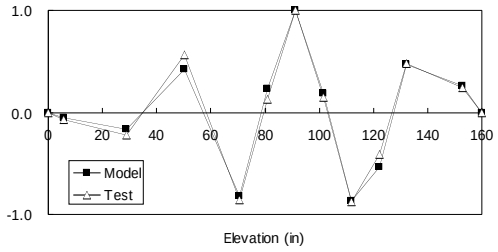
c) 3rd Mode



d) 4th Mode



e) 5th Mode



f) 6th Mode

Fig. 3 Comparison of Mode Shapes with Test Results

5.2 Static Analysis

The static analysis for the simplified fuel assembly model was performed to verify the static characteristics of the model. In the static analysis of the model, the central node of the model was deflected laterally to simulate the fuel assembly static test. The deflection shape of the model was compared with the static test. The static analysis result of the model is shown in Figure 4. The result of the model shows good agreement with fuel assembly test result.

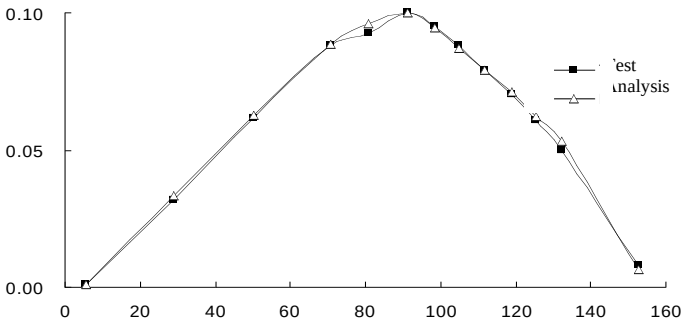
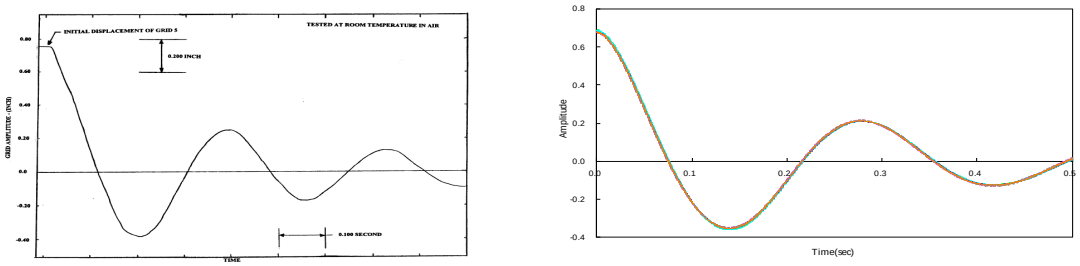


Fig. 4 Comparison of Static Analysis with Test Results

5.3 Pluck vibration Analysis

The pluck vibration analysis for the model was performed to verify the vibration properties of the model. In the pluck vibration analysis for the model, the central node of the model was deflected laterally and then released. From this analysis, the damping value of the model is determined to have equivalent damping behavior with the tested fuel assembly. The pluck vibration analysis result of the model is shown in Figure 5.



(a) Test (b) Analysis

Fig. 5 Comparison of Pluck Vibration Analysis with Test Results

5.4 Pluck Impact Analysis

The pluck impact analysis for the model was performed to verify the impact characteristics of the model. The contact elements between the grid node of model and wall were defined. The tested grid dynamic stiffness was used as the contact stiffness. In the pluck impact analysis for the model, the central node of the model was laterally deflected and then released. After the node is released, each node which is defined by contact element is impacted on the wall node which is imposed fixed boundary condition. Figure 6 shows the pluck impact analysis model. Table 2 shows the impact forces comparison for the test and analysis.

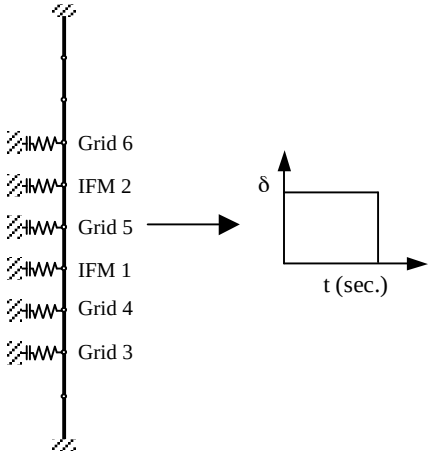


Fig. 6 Pluck Impact Analysis Model

Table 2 Comparison of Pluck Impact Force with Test Results (unit: lbs)

Items	Grid 3	Grid 4	IFM 1	Grid 5	IFM 2	Grid 6
Test	1255	1610	295	1325	-	910
Analysis	1431	1544	334	1517	284	1309

6. CONCLUSIONS

The simplified fuel assembly model for fuel assembly SSE and LOCA analysis has been developed with ANSYS code to evaluate the structural integrity of 17×17 RFA fuel assembly under SSE & LOCA events. The model has been developed using optimization technique. The model configuration and its design variables have been determined by sensitivity studies.

In order to verify the model, the simulations for static test, pluck vibration test, pluck impact test were performed. The optimized model has a good correlation with fuel assembly mechanical test results. In the near future, the final verification of the model and methodology should be performed by core analysis. It is necessary to study further evaluation for the other fuel assemblies and the core analysis.

References

H.K.Kim (2001). “Development of core seismic analysis models for KNGR fuel assemblies associate with 0.3g seismic loads” Nuclear Engineering and Design 212 (2002) 201-210

Vanderplaats, G.N (1984), “Numerical Optimization Techniques for Engineering Design: With Applications, McGraw Hill, N.Y.

Jeon, S.Y., Kim, I.K., Jeon, K.L. (1999), “Dynamic impact tests with spacer grids of PWR fuel assembly and impact performance in seismic analysis. Proceedings of the 15th SMiRT Conference

ANSYS Theory Manual