2017 Water Reactor Fuel Performance Meeting
September 10 (Sun) ~ 14 (Thu), 2017
Ramada Plaza Jeju • Jeju Island, Korea

PREDICTION OF 17x17 GRID CRUSH BEHAVIOR WITH A 2D FINITE ELEMENT MODEL

Jooyoung Ryu[1], Youngik Yoo[1], Namgyu Park[1], Jongsung Yoo[1]

[1] *KEPCO Nuclear Fuel: 242, Daedeok-daero 989beon-gil, Yuseong-gu, Daejeon, 305-353, Korea, and jyryu@knfc.co.kr, yiyoo@knfc.co.kr, nkpark@knfc.co.kr, jsyoo@knfc.co.kr*

ABSTRACT: *When the nuclear fuel is subjected to an unwanted load during shipping, handling or manufacturing, it could lead to fuel failure such as spacer grid buckling, cladding tube deformation. The cladding tube is important in that fission gas and nuclear materials can be isolated from the external environment. If the cladding is failed, leakage of nuclear materials are unavoidable. Therefore, consistent analytic damage prediction for evaluating spacer grid crush behavior as well as fuel cladding deformation is essentially required. For that reason, KNF has developed 2-dimensional model using ANSYS for the purpose of evaluating grid crush characteristic and fuel rod integrity under severe situations. Predictions from the 2-dimensional model showed it matches well with grid test results.*

KEYWORDS: *Nuclear Fuel, 17x17, Mid Grid, Static, Dynamic, Crush, Buckling, 2-dimensional (2D) Model*

I. INTRODUCTION

Buckling is induced from external short impact or gradual force during shipping and handling. In addition, lateral impact under seismic condition also could lead to grid buckling. Depending on the relative loading velocity, buckling can be divided into static buckling and dynamic buckling. Static buckling can be observed in the shipping container or manufacturing bench where the spacers are rigidly fixed by clamps. On the other hand, seismic/LOCA (Loss of Coolant Accident) condition induces dynamic impact between fuels that causes grid dynamic buckling. In addition, there exists the dynamic impact by dropping while fuels are handling in the fuel pool. If the unwanted accident is occurred, the fuels could have fatal damage. Especially, in case of spent fuels, it is weaker to impact load compared to fresh fuels.

The buckling behavior is nonlinear in nature. If the buckling behavior of many grid types is to be found, it needs to identify many parameters such as material properties, geometries, initial conditions and experiments, etc. Also, the spacer grid consists of a lot of straps with complex geometry. Fuel vendors usually depend on the grid buckling tests to identify buckling characteristics, but the test takes much budget as well as time (Ref. 1 & 2). Thus, finite element model (FEM) analysis can replace the test effectively, and once it establishes well it can estimate the buckling strength of a specific grid design. 3-dimensional model takes a long time to compute the dynamic or static behavior considering the huge size of the degrees-of-freedom. Therefore, this paper discusses the development of the 2-dimenional spacer grid model which will represent almost realistic grid characteristics. Detailed model generation using ANSYS 15.0 (Ref. 3) is presented, and the verification of the model is performed.

II. FINITE ELEMENT MODEL DEVELOPMENT

The target spacer grid is 17x17 mid grid assembly, and the basic concept of the 2-dimensional cell can be seen in Fig. 1. Beam elements are used to simplify grid straps and fuel tube. Since the fuel tube in the grid cell accompanies contact on the spring and dimple in the straps, a point contact element is used for contact simulation. The contact element can simulate the gap between the fuel rod and the spring or dimple, and the sliding between them can be simulated during buckling process.

The fuel tube configuration is successfully realized using the beam elements in the study. The concept of the plane model is very beneficial to identify many nonlinear behaviors including post bucking and further analysis.

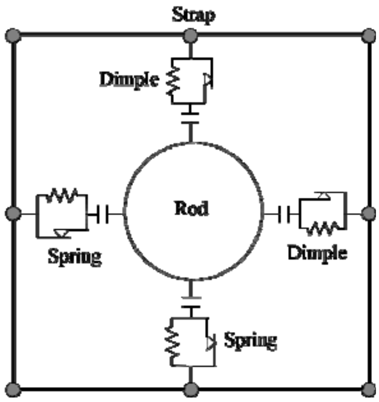


Fig. 1. Developed FE model concept of 1x1 grid

II.A. Procedure for the Model Development

The procedure for the 2-dimensional FEM generation can be summarized as listed in TABLE I.

TABLE I. Procedure for the 2-dimensional FEM

Items	Contents
Information for element definition	Element type, Properties of the elements, Geometries and dimensions, Material properties
Contact element generation	Definition of the target and contact surfaces (fuel tube to strap contact during post buckling) Definition of the point-to-point contact (fuel tube to spring/dimple contact during normal condition
Definition of the contact	Offset in the contact pair, Contact stiffness and penetration tolerance Normal direction of the contact surface, Sliding direction of the contact element Initial stress due to the interference between the spring/dimple and fuel tube Key option for the contact element
Boundary condition	Control method depending on the buckling process (displacement/force control)
Computation	Artificial damping coefficient, Number of sub-steps, Length of the strap Length of the cladding tube
Output	Static buckling: load-deflection curve, buckling strength, buckling mode, post buckling behavior, stress distribution on the cladding tube Dynamic buckling: load-deflection curve, buckling strength, buckling mode, post buckling behavior, stress distribution on the cladding tube
Comparison to the test results	Simulation, Experiment

II.B. Assumptions

A few assumptions are required to accomplish reduction of the computation time and confident outputs. It is also needed to simulate the grid buckling effectively. Thus, assumptions are defined as followings.

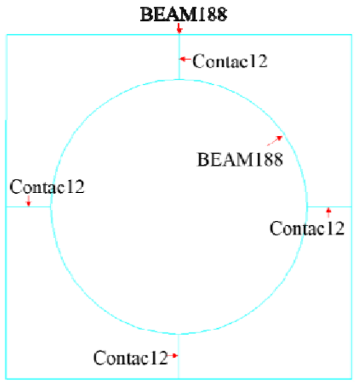
1) 2-dimensional beam elements are used for the grid straps and fuel cladding tube.
2) There are no cracks or damage in the structures, thus it has continuous geometry and uniform material properties.
3) No rotation is allowed in the cladding tube during deformation and sliding.
4) Pellets are neglected for simple calculation.
5) Guide tube sleeve is not included to simplify analysis.
6) Welding is needed to join the straps, and then the welding nuggets are produced as by-product. Welding joint can be implemented by using coupling the strap cross intersections.

7) Springs and dimples behave linearly.
8) Elastic modulus of the structure is constant.

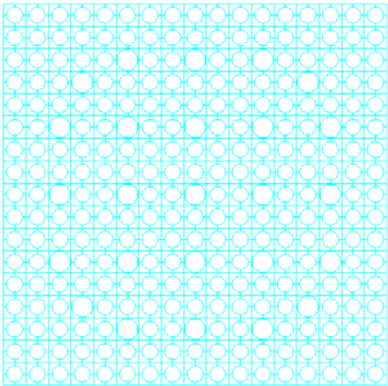
II.C. Finite Element Model Generation

Unit cell consists of 4 beam elements that represent straps and fuel tube as shown in Fig. 2 (a). The beam element for the strap and tube is BEAM188 in which the thickness and length can be implemented. BEAM188 which has 6 degrees-of-freedom is suitable for nonlinear analysis including large rotation and large deformation. By using arc length and nonlinear stabilization method, the model can simulate nonlinear buckling problem or collapse. In addition, BEAM188 supports material nonlinearity such as creep and plasticity.

There are 4 dimples and 2 springs in the unit cell in actual condition, but 2 dimples and 2 springs are used since the model is defined in the 2-dimensional space. CONTAC12 element is added to define contact geometry between the spring/dimple and fuel rod. Using CONTAC12, one can simulate sliding, friction as well as preload due to the initial interference. The geometric dimensions, physical properties for the material and the contact definition are based on the 17x17 mid grid (See Fig. 2 (b)) and the cladding tube that are made of zirconium alloy. And the nominal values of dimensions are applied.



(a) Unit cell



(b) 17x17 Grid

Fig. 2. Grid patterns of FE model

II.D. Boundary Conditions

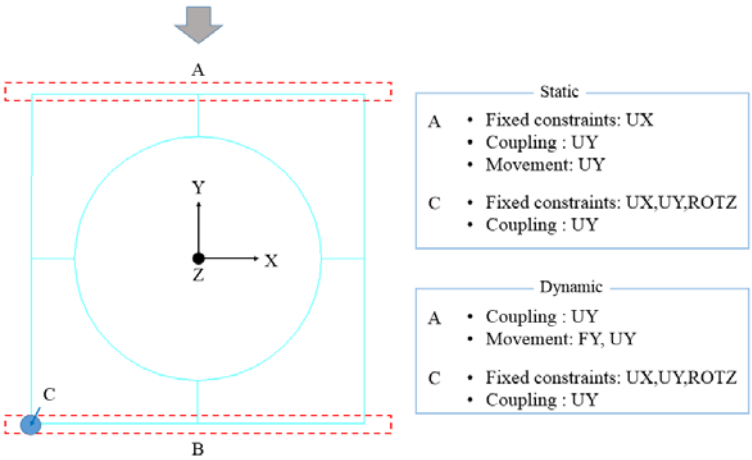
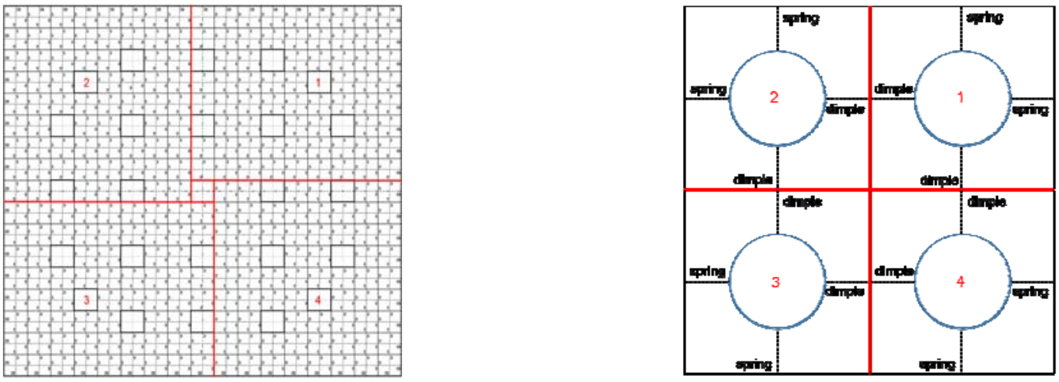


Fig. 3. Coupling and constraints condition

The constraints on the cell are visualized in Fig. 3. Since out-of-plane motions are not involved, x and y directional rotations and z-directional displacement are constrained. It is noted that all the nodes in the bottom line is fixed, since they are not allowed to move in any directions. Static buckling process computation can be realized by displacement control, the displacement constraint is applied on the master node in the top line. The displacement should be large enough so as to cause buckling and contact between the strap and the fuel rod. For dynamic buckling, two different control methods are used to simulate realistic impact force history. Impact force drastically reduces on the buckling, thus loading control leads to unrealistic results. That is knowing that the loading control has the reaction force followed the designated load, excessive deformation is not avoidable and succeeding computation must be required to reach to force equilibrium after contact between the straps and the fuel rod. Therefore, the loading control is used before buckling and the displacement control is applied after buckling.

II.E. Contact Strategy

Since the straps and the fuel tube are flexible media, the contact between them is flexible-to-flexible. TARGE169 and CONTA171 are utilized in the study, and the two elements are applicable to low order elements without no mid nodal points. The grid strap and the tube are generated using the beam element. Basically, it is the neutral surface in the beam element where the contact event occurs. For exact calculation of the contact, the properties in CONTA171 and TARGE169 were given considering the thickness of the beam element. CONTA12 can simulate contact and sliding between the fuel rod and spring/dimple. 17x17 grid is divided into 4 sections as shown in Fig. 4. The configuration in one-quater section exactly coincides to the next one with 90º rotation.



(a) Quad sections in 17x17 grid (b) Spring/dimple configuration in 17x17 grid

Fig. 4. Spring/dimple configuration for 17x17 array

II.F. Computation Method

Convergence difficulty due to an unstable problem is usually the result of a large displacement for smaller load increments. Nonlinear stabilization can be understood as adding an artificial damper or dashpot element at each node of an element that supports this technique (see Fig. 5). For the DOFs that are stable, the dashpot elements have little effect on the results because the displacements and the stabilization forces are small relative to the physical forces. The coefficient used to calculate the damping (stabilization) force is the damping factor. Although it has the same physical meaning and unit as physical damping, it is purely numerical in nonlinear stabilization. The program calculates a damping factor based on the energy dissipation ratio that one specify, or one can input the damping factor value directly. Therefore equilibrium equation in the structure turns to Eq. (1).

$$P - I - Fv = 0 \qquad (1)$$

Where, P = *External force*, I = *Internal force*, F_v*(Viscous forces)* = $cM^{*}v$, c = *Damping factor(input variable)*, v = *Nodal velocity*, M^{*} = *Artificial mass matrix defined by unit density*

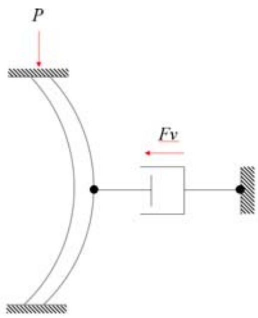


Fig. 5. Nonlinear stabilization method

III. VERIFICATION RESULTS

III.A. Static Buckling

The conceptual static buckling load-deflection curve can be seen in Fig. 6. Region A denotes almost linear elastic behavior before buckling. Region B corresponds to the buckling moment and post buckling response. For large deflection exceeding region B develops contact between the strap and fuel tube. The analysis results using finite element grid models are compared to the test results. The maximum tube stress intensity for large deflection was reviewed since the rod failure could happen under the condition in region C.

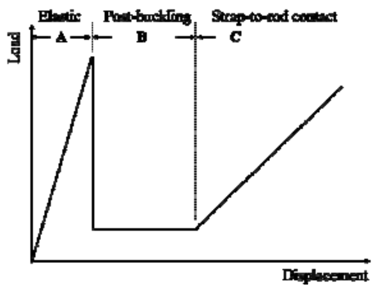
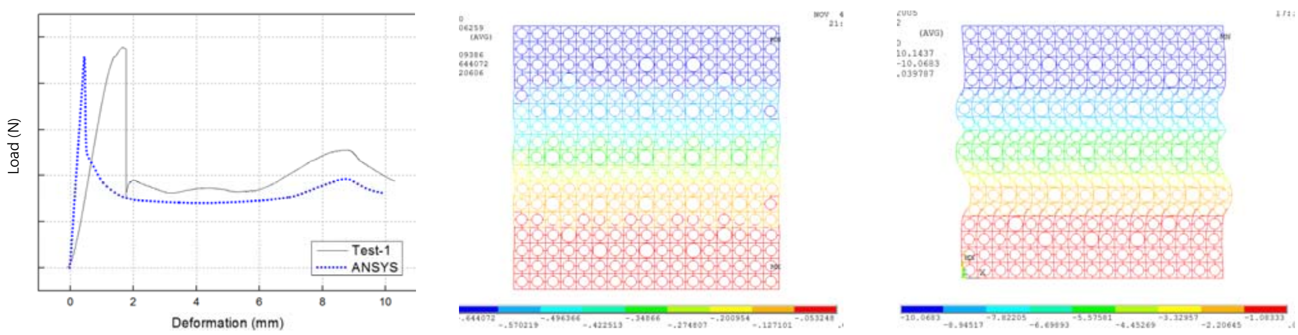


Fig. 6. Load-displacement schematic behavior in static analysis

Load-deflection curves, deformations and stresses for 17x17 grid models are delineated in Fig. 7. As shown in the Fig. 7 (a), the curves are quite similar to the test results. That is, the buckling force as well as the contact force resemble test results.

The buckling shapes are shown in Fig. 7 (b) and (c). Buckling deformation can be seen in Fig. 7 (b), and it can be observed that the buckling accompanies sliding. Also, it can be seen that the straps contact the fuel rod in Fig. 7 (c), and this state has drastically increased the load due to the contacts.



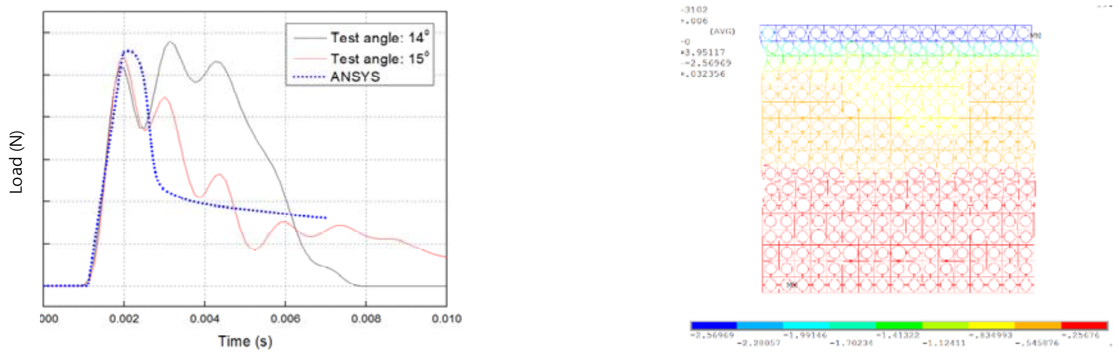
(a) Load-deformation behavior (b) Buckled shape results (c) Strap-to-rod contact after buckling

Fig. 7. Static analysis results

III.B. Dynamic Buckling

The model in the previous static buckling analysis is used for dynamic buckling analysis since it is geometrically identical. Dynamic buckling is a branch of transient phenomena so that ANSYS transient solution can be used. Vertical motion in the bottom strap is fixed, and the displacement control is utilized after the buckling. During the loading control process, it is controlled that the external loading is terminated at the buckling force in the grid test. In the real test, since the impact force disappears after contact, the loading control is applied at first and the displacement control is followed after buckling to simulate real condition.

The same artificial damping factor and substep number used for the previous static buckling analysis, and the transient analyses are applied to 17x17 grid dynamic buckling. The buckling behaviors are delineated in Fig. 8. Impact force histories from the analysis are close to the test results, thus the model and its parameters are reasonable.



(a) Load-time curves (b) Buckled shape results

Fig. 8. Dynamic analysis results

IV. CONCLUSIONS

This study describes the results on the 17x17 grid finite element model for buckling analysis. Because the buckling behavior is nonlinear in nature, it needs to identify many parameters such as material properties, geometries, initial conditions, etc. On the other hand, the spacer grid consists of a lot of straps with complex geometry. Therefore analytic approaches are very hard to implement. Finite element analysis can replace the test effectively, and it can estimate buckling strength of the specific grid design.

Using the commercial software, ANSYS, the 2-dimensional grid models are developed. Their performances are verified comparing to the test results. It is certified that the proposed model shows similar results with that of the test such as buckling behavior. In addition, although the true value is not identified, the fuel tube stress can be estimated. The developed model will be used for the structural integrity evaluation.

ACKNOWLEDGMENTS

This research is one of the results that have been supported by a research project at KNF. The authors would like to give thanks for the support of the participants who were involved.

REFERENCES

1. Thomas L. Sanders et al., “A Method for Determining the Spent-Fuel Contribution to Transport Cask Containment Requirements,” SANDIA REPORT, 1992
2. Wei Zhao et al., “LS-Dyna Application in Simulating Impact Tests of Nuclear Fuel Spacer Grids and Drop Tests of Fuel Shipping Package,” 12th International LS-Dyna Users Conference, 2012
3. ANSYS 15.0 Mechanical APDL Documentation