Transactions, **SMiRT-25**
Charlotte, NC, USA, August 4-9, 2019
Division V

A FINITE ELEMENT MODEL FOR NHR200-II 9x9 FUEL ASSEMBLY IN DYNAMIC ANALYSIS

Xicheng Wang[1], Dingqu Wang[2,5], Yueyuan Jiang[3,5], Songyang Li[4,5],

[1] Master Student, Institute of Nuclear and New Energy Technology, Tsinghua University, China
[2] Associate Professor, Institute of Nuclear and New Energy Technology, Tsinghua University, China
[3] Professor, Institute of Nuclear and New Energy Technology, Tsinghua University, China
[4] Assistant Professor, Institute of Nuclear and New Energy Technology, Tsinghua University, China
[5] Collaborative Innovation Center of Advanced Nuclear Energy Technology, Tsinghua University, China

ABSTRACT

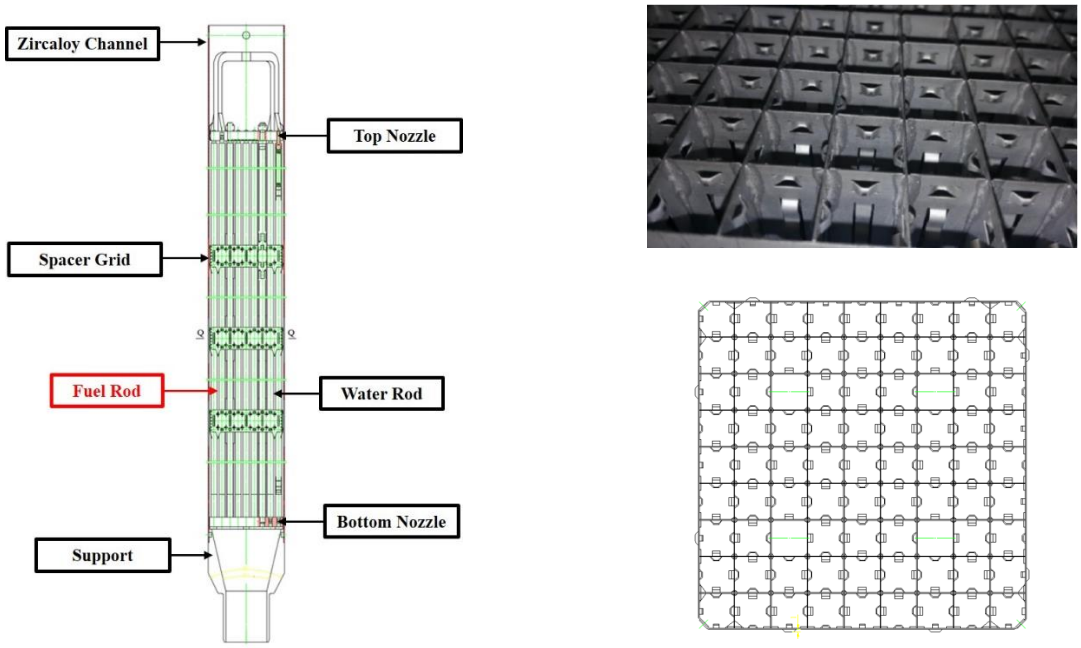
NHR200-II is a kind of nuclear heating reactor designed by Institute of Nuclear and New Energy Technology, Tsinghua University. The fuel assembly (FA) is similar to PWRs but with a zircaloy channel. The purpose of this paper is to propose a detailed finite element (FE) model of NHR200-II FA which can be used in the dynamic analysis (seismic, impact, etc). This FE model is modelled by using commercial FE software ABAQUS. The modelling techniques of three interaction of grids and fuel rods, channel and girds, channel and top & bottom nozzles are presented to provide the possible way to represent the behaviour of FA. Modal analysis and seismic simulation are performed to study the applicability of this FE model and to investigate its integrity during the earthquake. The results and discussing suggest that current FE model is sufficient to predict the dynamic response of NHR200-II FA in future research.

INTRODUCTION

Nowadays in China, especially in the winter, the weather becomes worse and worse since the massive use of coal for the heating system. It is necessary to find a new way to address such a problem. Therefore, NHR200-Ⅱ, a small modular reactor (SMR), aiming to supply enough heating source to surroundings, is designed by the Institute of Nuclear and New Energy Technology (INET), Tsinghua University. Its integrated layout, natural circulation, self-stabilization, non-active waste discharge and passive safety system ensure the inherent safety feature during its long operating time.

Earthquake and loss of coolant accidents (LOCA) can cause serious consequences on the core of reactor. Furthermore, the external force such as explosion load and crash load can damage the core in a short time. The fuel assembly (FA) is the core of a reactor, its structural integrity should be ensured in any accidents to allow the insert of the control rods to shut down the reactor. The FA of NHR200-II is similar to that of typical PWRbut with a zircaloy channel outside. It consists of the fuel rods, water rods, spacer grids, nozzles and channels, as shown in Figure 1.

The purpose of this paper is to propose a detailed finite element (FE) model of NHR200-II FA which can be used to study its structural behaviour induced by the dynamic load (seismic, impact, etc). The paper is organized as follows, Section 2 presents the model techniques of three different interaction between the grids and fuel rods, channel and grids, channel and top & bottom nozzles, Section 3 presents the dynamic analysis (modal and seismic) and the discussion of results.



(a) (b)

Figure 1. Schematic diagram of the fuel assembly of NHR200-II. (a) Whole model and (b) spacer grid.

3D FE MODEL OF FUEL ASSEMBLY

To capture the transient behavior induced by dynamic load and optimize the design of fuel assembly, a 3D finite element model of NHR200-II FA is developed. The detailed geometry is needed to study the (i) contact between the outside dimple of spacer grid and channel, (ii) integrity of the grids and fuel rods, (iii) interaction between the bottom nozzle and support as shown in Figure 1(a).

In this section, an overview of NHR200-II FA finite element model is summarized firstly. The connectors used to substitute the three-arc spring and dimple inside the spacer grid and the spring fixed in the top nozzle is introduced in the second part. The shell-solid coupling technique to connect the outside dimple and grid is presented in the third part. The interaction of the channel and top & bottom nozzles is described in the last part.

Overview of the FE Model

The FA of NHR200-II is made up of the channel subassembly (zircaloy-channel and support) and fuel bundle. The fuel bundle is typical PWRs type with 9x9-4 fuel rods, 4 water rods, 3 spacer grids, top and bottom nozzles. The 3D model developed by commercial FE software ABAQUS 6.14 is illustrated in Figure 2. Materials used in the simulation are listed in Table 1.

Fuel rods are modelled by beam element and the channel, grids, water rods are characterized by shell elements. The grid outside dimple, top, bottom nozzles and support are modelled with solid elements. Note that the top and bottom nozzle as well as support are set as the rigid body during the seismic simulation because of small deformation compared to the other parts. The pellets inside the fuel rods are not considered for simplicity. The thickness of rods Zircaloy-4 (Zr-4) cladding is increased to keep the same mass as the actual rods with pellets. The elastic-plastic model is used in Zr-4 property due to the possible yield during the simulation.

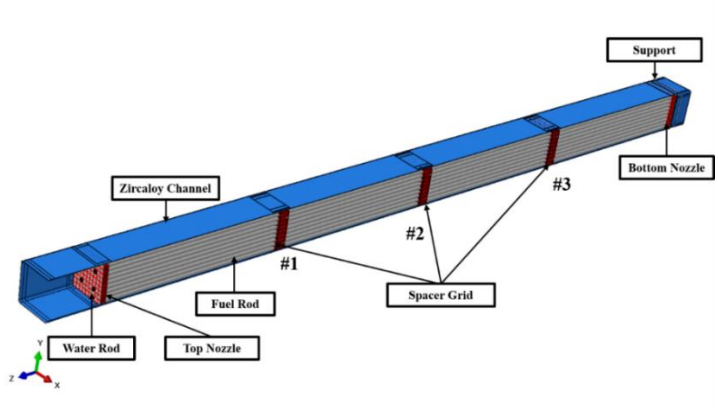


Figure 2. Fuel assembly of NHR200-II in Abaqus.

Table 1: Material properties of FA at room temperature

Name	Density $[kg/m^3]$	Young's Modulus $[Pa]$	Poisson's Ratio [-]	Used in
Zircaloy-4	6570	9.1E10	0.33	Channel, Fuel rods, Water rods
GH4169	8190	2.1E11	0.3	Top and bottom- nozzle, Spacer Grid, Support

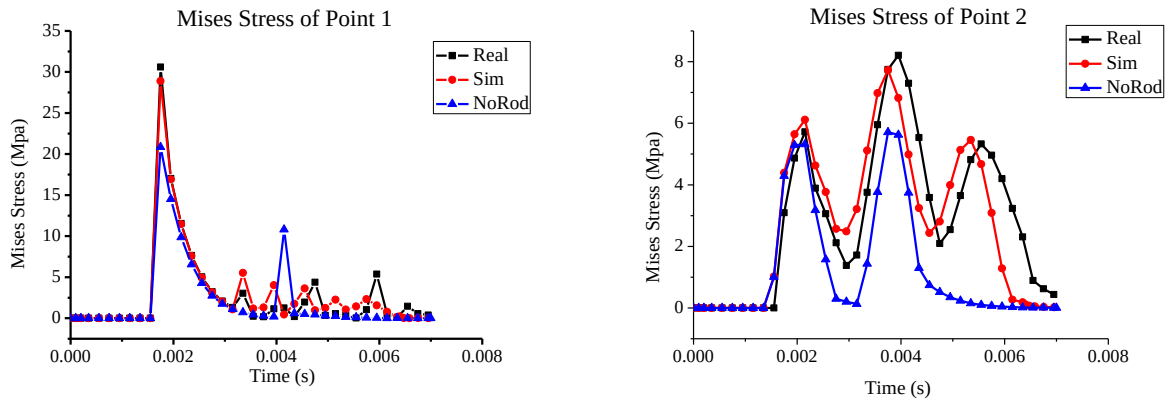
Interactions of Grids and Fuel Rods

One of the difficulties in modelling is the interactions between the grids and rods. For each rod, there are 6 contact pairs in each grid cells, as shown in Figure 1b. The total amount of contact pairs is too large to be simulated precisely. Connector elements (Abaqus 6.14) provide an easy and versatile way to model that and many other type of physical mechanisms whose geometry is discrete (i.e., node-to-node), yet the kinematic and kinetic relationships describing the connection are complex. Therefore, they are used in this model to substitute springs and dimples punched on the spacer grid to improve the computational efficiency.

Yoo et al. (2018) and Jiang et al. (2016) used the similar method to simplify the contact between rod and grid. However, there isn't a channel model for the FA of PWR. Lin (2007) also took the same approach for her NHR200-I model with the channel, but the nonlinear of connectors and contacts between spacer grids and the channel were not taken into consideration.

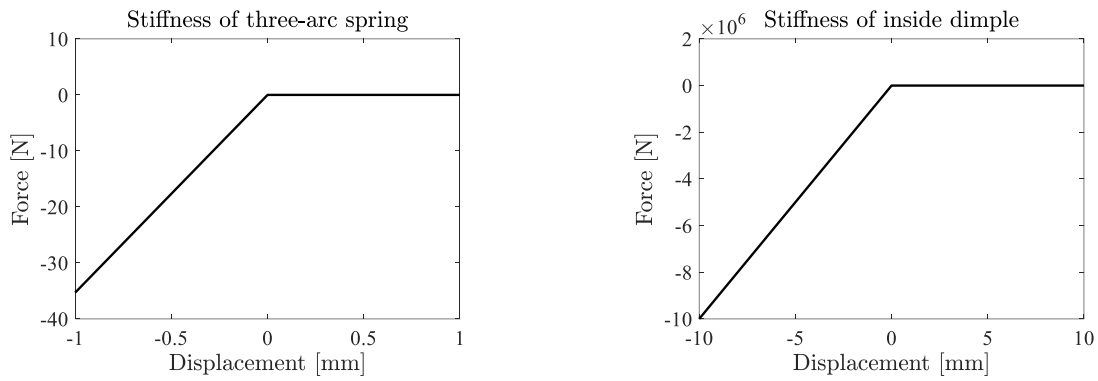
The previous study by Wang et al. (2018) demonstrates the feasibility of using connectors to replace the three-arc springs and inside dimple during the dynamic analysis, shown in Figure 3. Note that the 'real' presents the original model and the 'sim' is the simplified model with connectors. The 'no rod' indicates the skeleton model without the fuel rods. The large variation of 'NoRod' and 'Sim' curves suggest that the connectors are able to reveal the effect of fuel rods.

In this paper, the connectors can be divided into two types of non-linear axial-spring with different stiffness (Figure 4). The stiffness of the three-arc spring and the inside dimple are from the work done by Shen (2013) and Abaqus default rigid value, respectively. The negative force and displacement indicate that the connectors are compression-only spring. The non-linear part of three-arc spring is not included due to the small displacement in the seismic calculation. The top view of the mid-gird can be seen in Figure 5.



(a) (b)

Figure 3. Evolution of Mises stress of channel induced by an impact load from Wang et al. (2018) at the contact area of (a) up-grid #1 and (b) mid-grid #2 (Figure 2).



(a) (b)

Figure 4 Stiffness of (a) three-arc spring and (b) inside dimple used in Abaqus

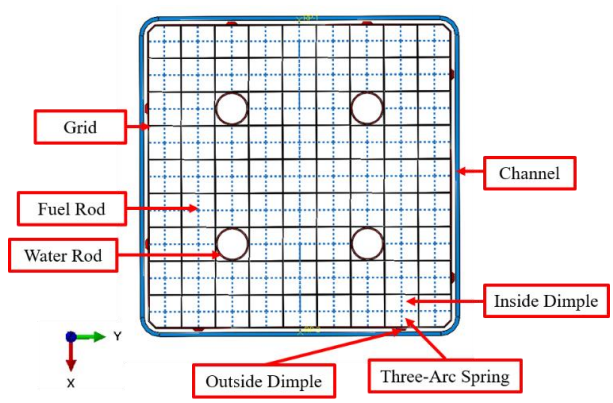


Figure 5. Top view of the mid-gird using connectors

Interactions of Channel and Grids

The grid outside dimple used to avoid direct contact between the channel and grid can be seen in Figure 6. There exists a 0.8mm clearance between the channel and grids, as shown in Figure 5. For simplicity, the grid is modelled with the shell element (S4R). However, the grid outside dimple which is an important part should be created by the solid element (C3D8I) to study its integrity and stress distribution during the dynamic analysis.

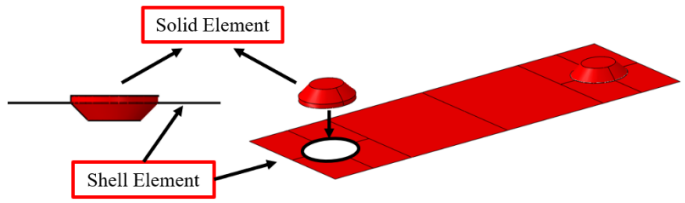
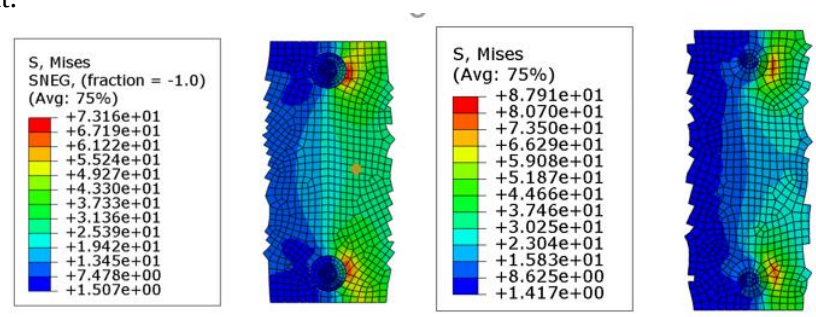


Figure 6. diagrammatic sketch of grid outside dimple and gird

The shell-to-solid coupling is a feature in Abaqus by which 3D shell element can be coupled automatically to the 3D solid element. The simulation results illustrated in Figure 7 demonstrate that the shell-to-solid coupling technique can effectively produce a similar result compared to the cases with the total solid element.

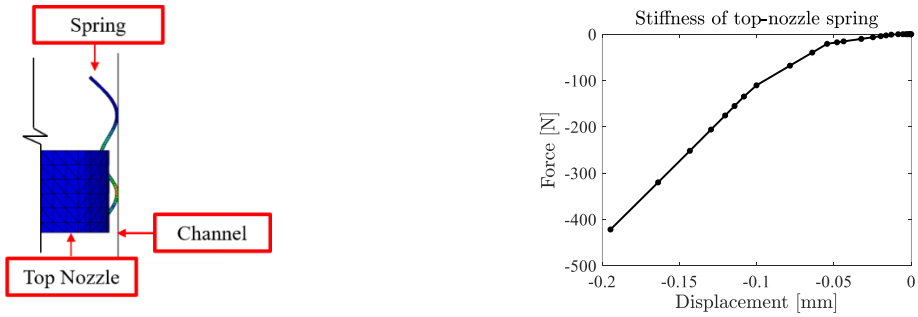


(a) (b)

Figure 7. Mises stress induced by 2mm displacement on outside dimple of (a) solid element of dimple and shell element of the grid and (b) solid element of dimple and grid

Interactions of Channel and Top & Bottom Nozzles

Eight special designed springs are fixed on the top nozzle and contact directly to the channel to mitigate the vibration induced by the dynamic load. The connectors discussed in the interaction of girds and fuel rods are applied to take the place of springs. Simulation of the spring and channel contact was done to study the characteristic of spring since there are no available results for measurement of the stiffness of spring, shown in Figure 8.



(a) (b)

Figure 8. (a) Original model with spring before the simplification and (b) the total stiffness of two springs in one side used in connectors

The bottom nozzle is sat on the support directly and surrounded by the channel with 8 mm gap as shown in Figure 9. A surface-based tie constraint (Abaqus 6.14) is used to tie the channel and support as well as fuel rods and bottom nozzle during the simulation. A non-friction surface-surface contact is used between the channel and bottom nozzle. A friction surface-surface contact with frictional coefficient of 0.15 is created between the bottom nozzle and support.

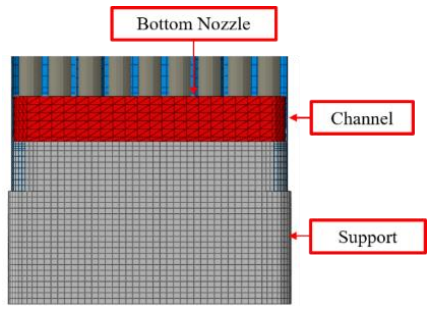


Figure 9. Geometry of channel, support and bottom nozzle.

DYNAMIC ANALYSIS AND RESULTS DISCUSSION

In this section, the modal and seismic analyses are performed to investigate whether the currently available FE codes can be suitably used to study the response of NHR200-II fuel assembly under the dynamic load. The top & bottom nozzles and the support are assumed as the rigid body in this section since they are not the concerned structural compared to the other parts.

The modal analysis was conducted with the fuel bundle model without the channel and support to study the dynamic characteristics including the natural frequency and modal shape. An earthquake spectrum of the magnitude of 6.5 on the Richter scale was used in the seismic simulation to investigate the integrity of the FA. A sensitivity study was carried out to investigate the effect of the friction coefficient and stiffness of top-nozzle spring on the contact force between the grid outside dimple and channel.

Modal Analysis

The modal analysis was conducted with the fuel bundle. The reference points of two nozzles (top and bottom) were fixed as the boundary condition. The first three principle modes frequency in X direction (Figure 2) are listed in Table 2, which are quite small compared to the frequency of the channel (more than 250 Hz, Lin (2007)). The effective mass of these modes is 90 kg, which accounts for 82% of the total mass of fuel bundle (110 kg). Figure 10 presents the shapes of the above modes with a 200 deformation scale factor. The typical FA of PWR mode shapes with the 7 and 8 girds are shown in Figure 11, respectively.

Table 2: Natural frequency of principle modes in X direction

Mode number	1	2	3
Natural Frequency (Hz)	4.787	12.166	22.969

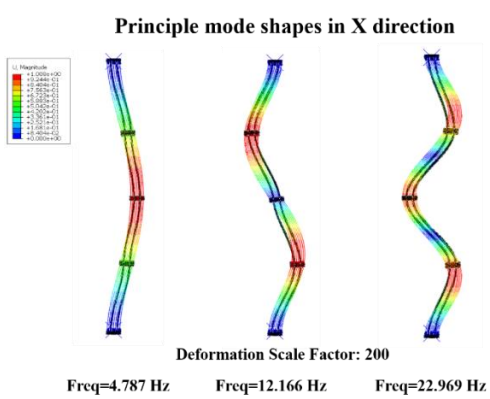
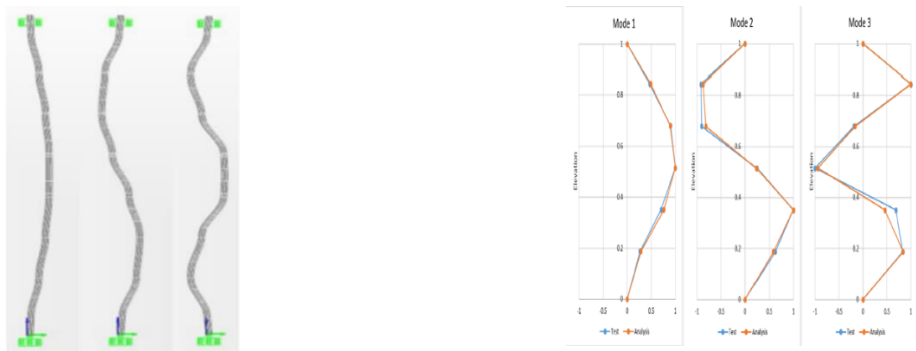


Figure 10. Principle mode shapes in X direction from simulation results



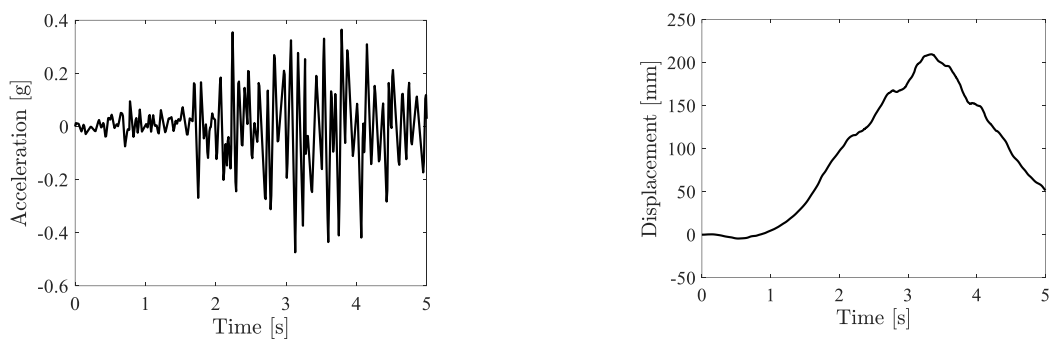
(a) (b)

Figure 11. Typical PWR's fuel assembly mode shapes from (a) Weichselbaum et al. (2015) and (b) Yoo et al. (2018)

The large difference of natural frequency between the fuel bundle and channel suggests that the channel will remain stationary relative to fuel bundle during the dynamic analysis. The vibration mode of NHR200-II FA in X direction (the same as Y direction due to the symmetry) is mainly superposed by three principle bending modes. The comparison among the typical PWR's FA and the results from simulations validate that this 3D FE model could represent the dynamic characteristics of the prototype model.

Seismic Analysis

In this section, we consider a simulation of the NHR200-II FA, which was subjected to an earthquake of magnitude 6.5 on the Richter scale. The seismic load which was horizontally (X-direction in Figure 2) added at the reference point of the support (top nozzle, bottom nozzle and support were simplified as the rigid body in seismic analysis) and the displacement of support are presented in Figure 12, respectively. The simulation was performed to investigate the predictability of the FE model of NHR200-II and to assess the structural integrity of the practical NHR200-II FA.



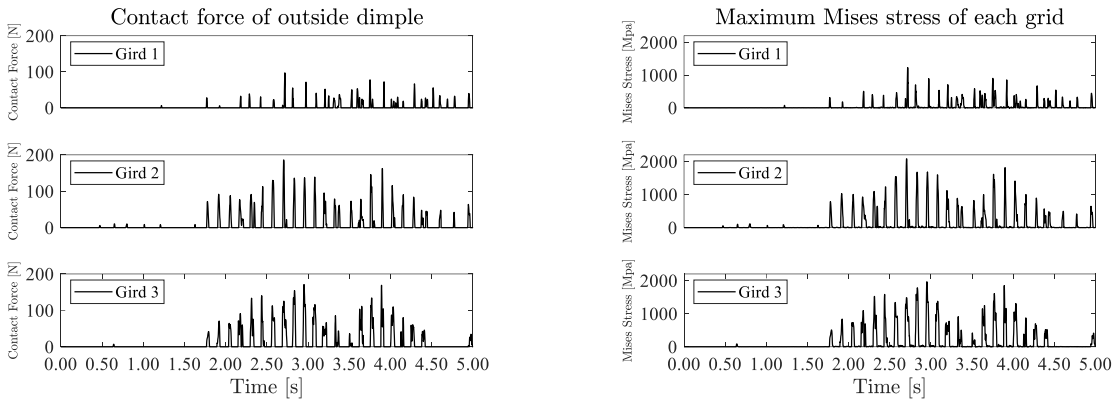
(a) (b)

Figure 12. (a) Ground acceleration records from Abaqus 6.14 and (b) displacement of support

1. ***Outside dimple and grid***

The total force due to contact pressure of dimple welded on the grid outside surface are presented in Figure 13 (a). It is noted that for each grid, only one outside dimple resutls, shown in Figure 14 (red circle), is selected. The maximum Mises stress from the ambient mesh of above outside dimple (e.g. for grid 2, a shell mesh of grid in red circle with maximum mises stress diplaed by orange is selected) are depicted in Figure 13 (b). In these figures, we can find that the three most violent collisions were captured at t=2.7s, t=2.95s and t=3.9s, respectively. The stress distribution contour of grid 2 at t=3.9s is shown in Figure 14 (the field variable output at t=2.7s was not captured due to the large output frequency).

The Mises stress of gird 2 is far beyond the yield stress of GH4169 alloy which means that the elastic-plastic model of this material is needed in the future simulation. Since there is no experiment for the mechanical performance of this new designed spacer grid, it is hard to decide a proper modelling way to represent its structural behavior. Therefore, more experiment research is needed to study its integrity especially at the interaction area between the outside dimple and gird.



(a) (b)

Figure 13. (a) Contact force of outside dimple on each grid and (b) maximum Mises stress of each grid

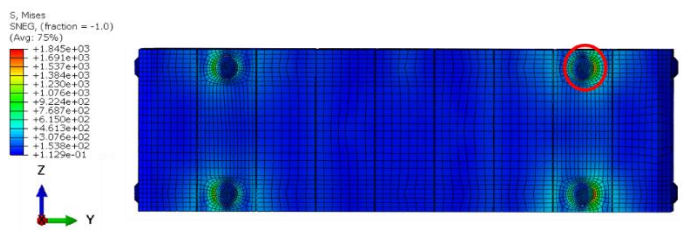
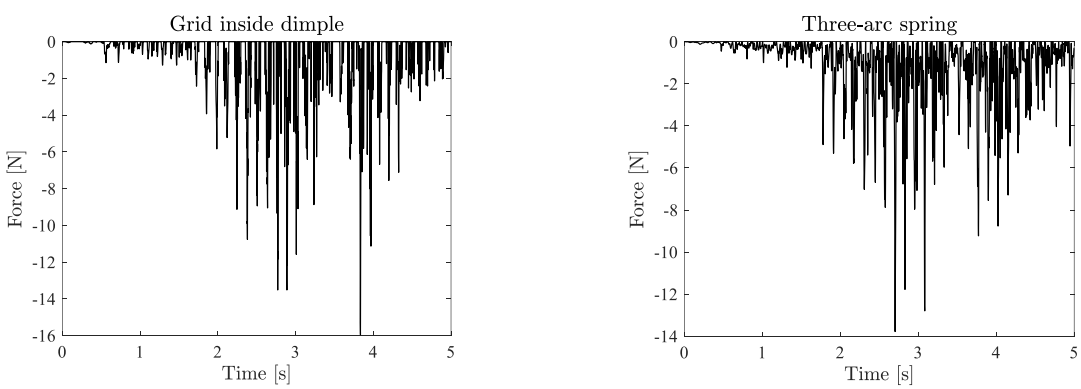


Figure 14. Mises stress of Grid 2 at t=3.9 s

Figure 15 shows the connectors force of three-arc spring and grid inside dimple (Figure 5). All the force of two connectors fall in its pre-designed range (Figure 4) indicates that there was no large relative displacement between the fuel rods and grid during the simulation. Meanwhile, the previous study presented in Figure 3 shows that the connectors can reveal the actual interaction of fuel rods and grid to some degree. Hence, it is possible to simplify the real three-arc spring and inside dimple by using connectors. However, in other types of the dynamic analysis (impact or explosive load), the stiffness of connectors should be paying more attention in terms of the non-linear part because of the huge contact force and slip caused by the severe collision among these two objects in the practical situation.

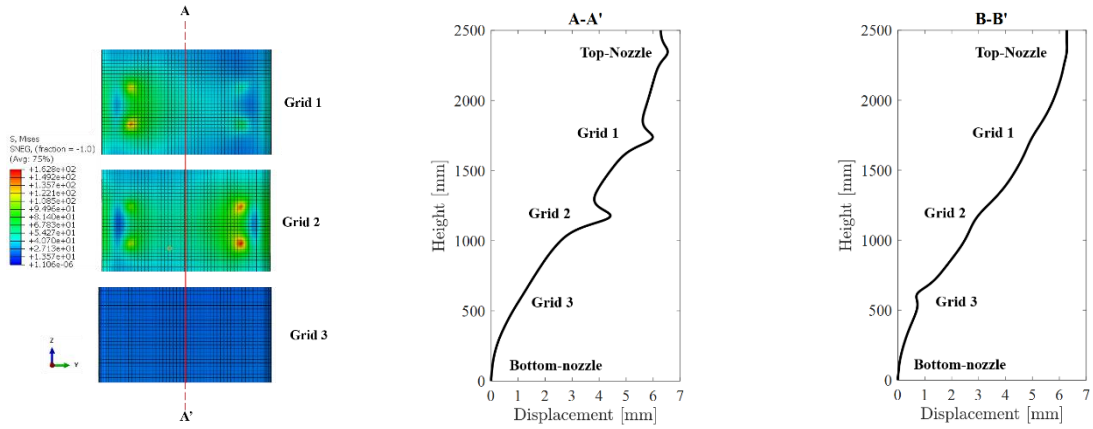


(a) (b)

Figure 15. Connectors force of (a) grid inside dimple and (b) three-arc spring in grid 2 (Figure 5).

2. *Channel*

Figure 16(a) shows the stress distribution contour of the channel on the contact area with the grid at t=3.9s. The maximum Mises stress of the channel is far below the yield stress of Zr-4 which demonstrates that the channel would be safe during the seismic analysis. The relative displacement of the channel surface centreline (A-A' and B-B') in load direction are presented in Figure 16(b) and Figure 16(c). Note that the zero displacement points for A-A' and B-B' are placed in the bottom of the channel separately. Accurate prediction of the channel relative displacement is important to ensure the successful insert of the control plate at any time during the accident. The maximum relative displacement of the channel at t=3.9s is 6 mm which gives enough space for the control plate.

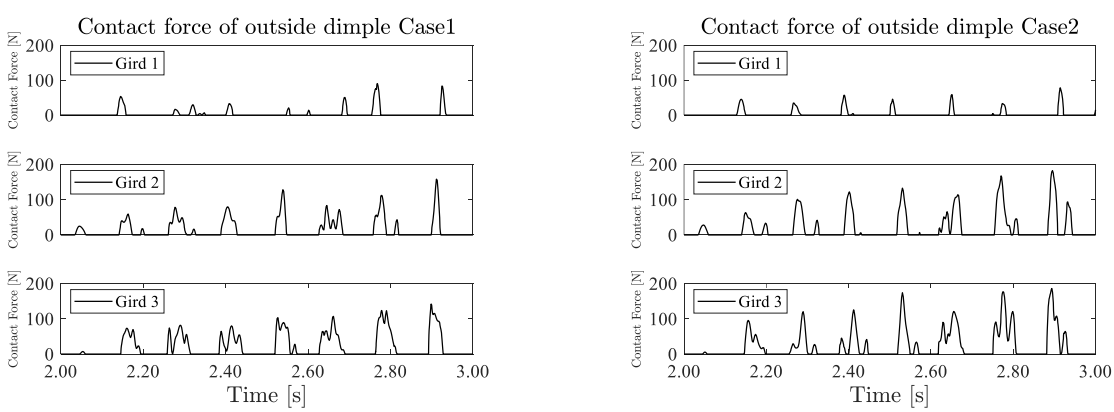


(a) (b) (c)

Figure 16. (a) Mises stress of channel at the grid height and (b) relative displacement of channel centreline (A-A') in the positive X direction and (c) relative displacement of channel centreline (B-B' not labelled) in the negative X direction at t=3.9s.

3. *Sensitivity study*

Two seismic simulations with partial seismic load (2-3s from Figure 12(a)) were performed to study the sensitivity of top-nozzle spring stiffness and the friction coefficient between the bottom-nozzle and support. For case 1, only the friction coefficient was changed from 0.15 to 0.05 compared to the previous case. For case 2, the friction coefficient was changed from 0.15 to 0.3 and the stiffness of top-nozzle spring was set as rigid, respectively. The total force of two cases due to contact pressure of outside dimple at the same position shown in Figure 14 (red circle) are presented in Figure 17.



(a) (b)

Figure 17. Contact force of outside dimple on each grid for (a) case 1 (friction coefficient of 0.05) and (b) case 2 (friction coefficient of 0.3 with rigid top-nozzle spring).

From Figure 17, we can observe that the outside dimple contact force of case 1 is generally smaller than that of case 2, especially at the grid 3. This is because of the different connection characteristics between the two cases. Case 2, with a large friction coefficient and rigid top-nozzle spring, has a tighter connection than case 1. A tighter connection means that more energy will be transferred from support to fuel bundle during the earthquake. However, the strategy of how to mitigate the seismic response can't be derived from analysing these two simple cases. Therefore, deeper reason of the effect of coefficient and spring stiffness and the specific mitigation strategy which are beyond the scope of this paper will not be discussed.

CONCLUSION

A detailed finite element (FE) model of NHR200-II FA used in the dynamic analysis (seismic, impact, etc) was introduced in this paper. The modelling techniques of three interaction of grids and fuel rods, channel and girds, channel and top & bottom nozzles were presented to give the effective way to simulate the dynamic behaviour of FA. Modal analysis and seismic simulation were performed to study the applicability of this FE model and to investigate its integrity during the earthquake.

Good agreement of modal shape with the NHR200-II FA and typical PWR's FA shows the validity of the FE model. The extremely large Mises stress at the interaction area between the outside dimple and grids suggests that further experiments should be done to calibrate the shell-coupling technique. Evolution of contact force and connector force demonstrates the feasibility to simplify the grid inside dimple and three-arc spring by using connectors. The 6mm maximum relative displacement of the channel ensures the successful insert of control rods during the 6.5 magnitude earthquake. The sensitivity study indicates that the contact force between the grids and channel can be mitigated by adjusting the friction coefficient and stiffness of top-nozzle spring. However, more studies should be done to obtain detailed approaches to mitigate the seismic response. In summary, the NHR200-II FA finite element model, with accurate and computational efficiency characteristics, can be used to predict the dynamic behaviour induced by variant loads.

The extension of this NHR200-II FA finite element model in terms of the transient simulation is possible in future work. However, more attention should be paid regarding the stiffness of connectors and the elastic-plastic model of channel and grid materials due to the high strain rate characteristic of this kind of problem.

REFERENCES

Abaqus 6.14. ABAQUS analysis user's manual, Dassault System, 2014.

Jiang, H., Wang, J, J. (2016). "Spent Nuclear Fuel System Dynamic Stability under Normal Conditions of Transportation," *Nuclear Engineering and Design*, UK, 310 1-14.

Lin, L. (2007). "Dynamic Responses Analysis of Fuel Assembly of Nuclear Heating Reactor (NHR) under Impact Load", Master thesis, Tsinghua University, Beijing, China.

Shen, T. (2013). "Study and experiment research on the mechanical performance of the spacer grid of NHR-II fuel assembly", Master thesis, Tsinghua University, Beijing, China

Wang, X, C, Wang, D, Q etc. (2018). "The simplified modelling method of dynamic analysis for fuel assembly grid", The 1st Chinese Nuclear Engineering Meeting on Structure Mechanics. China, Chengdu.

Weichselbaum, N, A, Abkenar, M, R etc. (2015). "Validation Facility and Model Development for Nuclear Fuel Assembly Response to Seismic Loading", *Journal of Nuclear Engineering and Radiation Science*, ASME, 041005-1

Yoo, Y, Kim, K, et al. (2018). "Finite Element Analysis for Fuel Assembly Structural Behavior," *ICONE26*. London, England.