Transactions of the SMiRT 27
Yokohama, Japan, March 3-8, 2024
Division IX *(include assigned division number from I to XII and remove this blue explanation text)*

FINITE ELEMENT SIMULATION ANALYSIS STUDY OF FUEL ASSEMBLY GRID ANTI-HOOKING PERFORMANCE VERIFICATION

Junhua Liu[1], Jingya Sun[2], Guoxun Zhang[3], Tang Yingzhuo[4] , Fujun Gan[5], Yunqing Zhou[6] , Zhaoxiong Dong[7]

[1]Assistant engineer, Shanghai Nuclear Engineering Research and Design Institute, Shanghai China (liujunhua@snerdi.com.cn)

[2]Senior engineer, Shanghai Nuclear Engineering Research and Design Institute, Shanghai China

[3]Assistant engineer, Shanghai Nuclear Engineering Research and Design Institute, Shanghai China

[4]Assistant engineer, Shanghai Nuclear Engineering Research and Design Institute, Shanghai China (tangyingzhuo@snerdi.com.cn)

[5]Professor level senior engineer, Shanghai Nuclear Engineering Research and Design Institute, Shanghai China

[6]Professor level senior engineer, Shanghai Nuclear Engineering Research and Design Institute, Shanghai China

[7]Senior engineer, Qinshan Nuclear power, Qinshan China

ABSTRACT

A numerical simulation model of the nuclear fuel assembly lattice of pressurized water reactor was established by using ABAQUS software. The process of local hooking of the lattice was simulated, and the anti-hooking performance of the improved fuel assembly lattice structure was analyzed. The comparison between the numerical simulation results and the experimental results verifies the rationality of the finite element analysis method established in this paper. The analysis method can be used for the analysis and evaluation of the anti-hooking performance of the fuel assembly lattice. There are two new types of lattice design in this study, and the process of hooking and hanging of lattice of different plan is analyzed, The results of the original lattice analysis were compared. At the same time, the impact of design improvement on key mechanical properties such as lattice clamping force and lattice spring stiffness was also evaluated. The results show that the design of improved lattice can effectively improve the anti-hooking performance of the lattice. In addition, the key mechanical properties such as clamping force and spring rate of the improved rear lattice frame are less different from those of the original scheme.

INTRODUCTION

Pressurized water reactors (PWR) nuclear fuel assemblies are prone to hooking accidents during loading and unloading, which will bring great safety risks and economic losses. Figure 1 shows the damage to the corner grids of the fuel assembly lattice during the refueling overhaul of a PWR nuclear power plant , The analysis results show that the phenomenon of hooking occurs in the process of hoisting of the fuel assembly, and the small chamfering angle of the four-corner grid structure of the fuel assembly lattice and the existence of small step surface are the main reasons for the hooking phenomenon of the lattice. In order to eliminate the risk of corner hooking in the original plan of the fuel assembly (as shown in Figure 2), two improvement design are given as shown in Figure 3. In order to screen the optimal plan in a short time and carry out experimental verification, it is necessary to evaluate the anti-hooking effect of the improved plan. Presently, the verification of the anti-hooking performance of the lattice is mainly based on test methods, however, the test piece needs to be processed and manufactured before the test can be carried out, which makes the verification period usually long. Therefore, this study was carried out based on the hooking test of fuel assemblies and the design of the improved lattice frame, and the hooking performance of the lattice was analyzed.

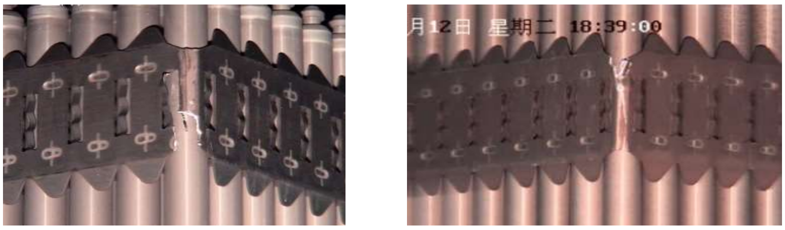


Figure 1. Damage to the grille elements of the fuel assembly lattice during the refueling overhaul of a nuclear power plant

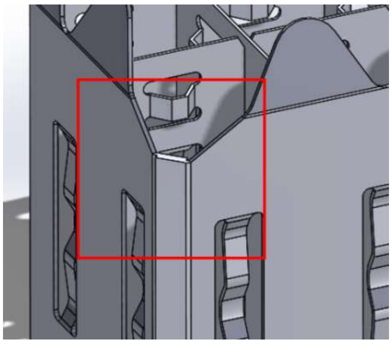
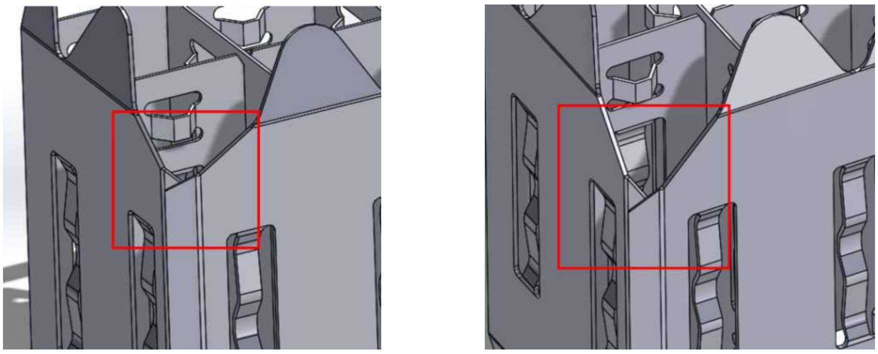


Figure 2. The original plan was designed at the corner position of the lattice



(a) Optimized paln A (b) Optimized plan B

Figure 3. Optimized palns was designed at the corner position of the lattice

HOOKING TEST OF FUEL ASSEMBLIES

The fuel assembly hooking test is used to verify that the improved anti-angle hooking design of the fuel assembly lattice can effectively reduce the risk of lattice hooking. The fuel assembly hooking test device is composed of a base steel plate, a rigid backplate that supports and fixes the fuel assembly, a transverse loading device, a simulated part of the core lower plate, and a lifting device for hoisting and moving the fuel assembly, etc., and its three-dimensional schematic diagram is shown in Figure 4. In the hooking-mounted test, a group of fuel assembly test pieces (fixed fuel assembly) are fixed on the test bench, and another set of fuel assembly test pieces (movable fuel assembly) are lifted and moved by a high-altitude crane, and a lateral load is applied on the movable fuel assembly (some working conditions do not need to apply lateral load), so that it is in close contact with the fixed fuel assembly at a certain angle and direction and friction occurs, and the actual hooking-mounted working conditions are simulated. Observe and record whether the movable fuel assembly shakes or abnormal sound during the lifting and lowering process, and observes whether the lattice is deformed or torn by the guide wing or corner grid element after the suspension test, and judges whether the lattice hooking or lattice damage caused by hooking occurs. There are three types of hooking tests for fuel assemblies, namely positive working conditions, dislocation working conditions and angular diagonal working conditions, and the hooking tests of unimproved lattice components are carried out in this study, and the test hooking results of some angular diagonal conditions are shown in Figure 5. The results show that under a small number of tests, the hooking phenomenon occurs between the fuel assemblies of the unimproved lattice frame, and the corner grid element is scratched due to the small step surface at the corner of the lattice angle grid, resulting in the damage of the lattice frame. In order to improve the design of the lattice frame, two design improvement plan as shown in Figure 3 are proposed. In order to verify the effect of the improved plan, the following finite element analysis was carried out.

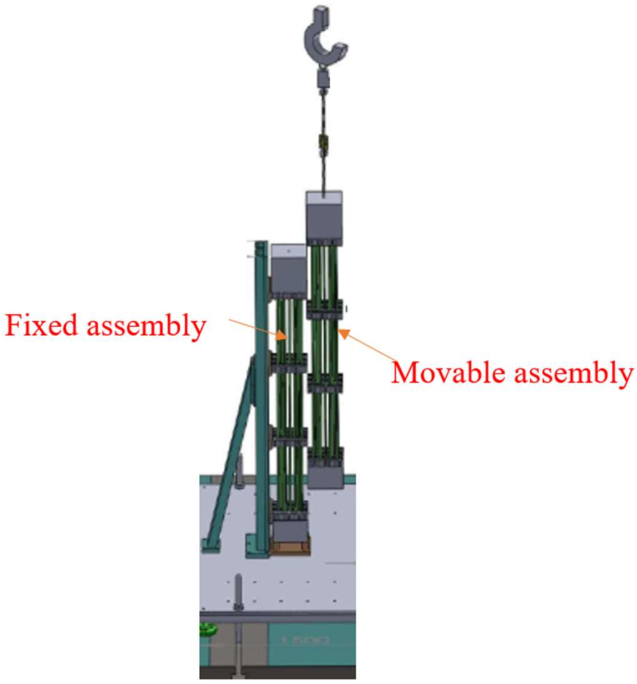
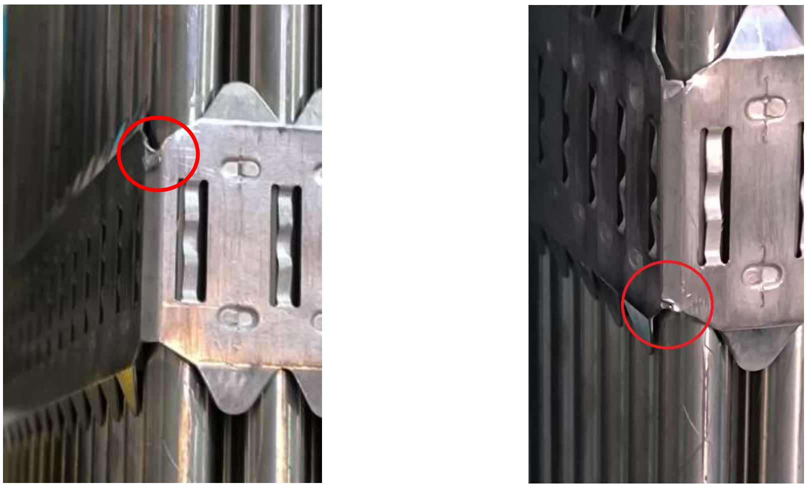


Figure 4. Hooking-mounted test device of fuel assmblies



(a) Lattice (Fixed assembly) (b) Lattice (Movable assembly)

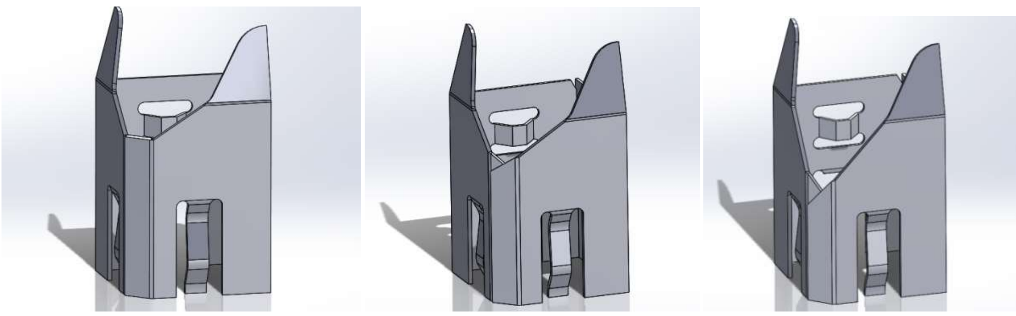
Figure 5. Lattice damage due to hooking of Fueel assembly in hooking test

FINITE ELEMENT ANALYSIS OF THE HOOKING PROCESS

Based on ABAQUS software, the mechanical properties of the improved lattice were analyzed, including the analysis of the lattice hooking and the stiffness of the lattice grid element, the purpose of which was to compare and evaluate the anti-hooking effect of the improved lattice and the stiffness analysis of the lattice element to verify that the change of the geometric structure of the lattice after the improvement would not have a significant impact on the stiffness and clamping force of the angular grid element. The 3D modeling software was used to establish and improve the three-dimensional geometric model of the frame, and the model was imported into the ABAQUS for finite element calculation.

Hooking analysis model

The lattice hooking hanging analysis was carried out for the unimproved lattice frame and the two improved lattice frames, respectively. The phenomenon of angular hooking of the fuel assembly is identified, half of the single corner grid element of the lattice frame is analyzed for hooking, and the geometric model of the unimproved lattice frame and the improved lattice is shown in Figure 6.

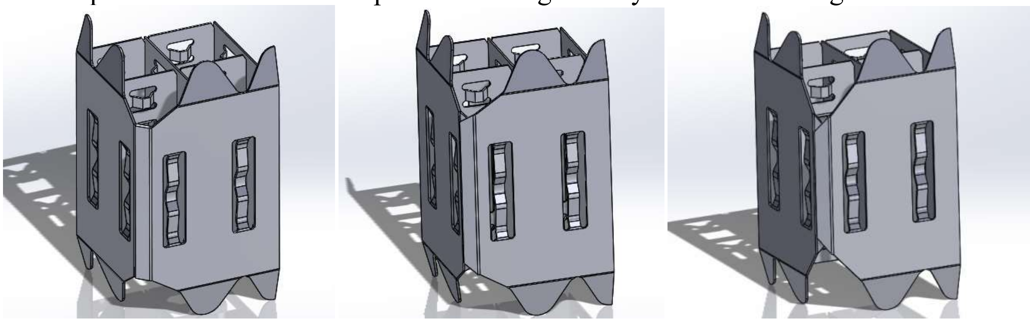


a) Original plan b) Plan A c) Plan B

Figure 6. Hooking analysis geometry

2 ）Lattice stiffness analysis model

In order to reduce the influence of boundary constraints on the calculation of the spring stiffness of the angular grid elements, four grid elements in the lattice were selected to establish the model. Wherein, the simplified unimproved lattice and the improved lattice geometry are shown in Figure 7.



a) Original plan b) Plan A c) Plan B

Figure 7. Lattice stiffness analysis geometry

Boundary conditions

1）Hooking analysis Boundary conditions

Taking the hooking analysis of the lattice without improvement as an example, the model of the lattice hooking analysis is shown in Figure 8, one of the grids (fixed angle grids) is fixed, a certain Y-direction displacement load is applied to the other gate (movable angle grids), and the Z-direction constraints and X, Y, and Z axial rotation constraints are applied at the same time. Since the fuel assembly (movable grid element assembly) may be deflected when the lattice is hookinged, a certain force is exerted on the fixed angle grid element assembly, which acts on the fixed gate element assembly in the X direction. In order to consider this force, the grid elements are connected by setting a certain stiffness of the X-direction spring, and the process of angular hooking is simulated, and the magnitude of the force between the grid elements is conservatively given with reference to the lateral load of 150N under the working condition in the lattice hooking test. In the hooking analysis of different plan, except for the different models, the other constraints and loading conditions are the same way.

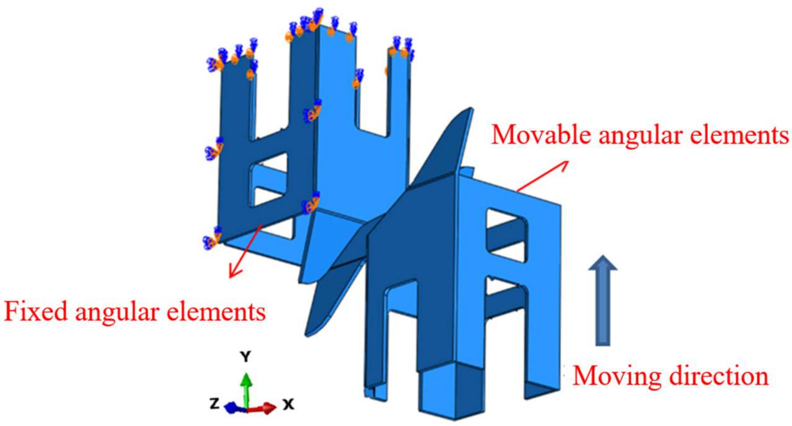


Figure 8 The assemble figure of lattice hooking analysis

2）Lattice stiffness analysis Boundary conditions

Taking the calculation of the stiffness of the grid element without improvement as an example, the model of the grid element is established as shown in Figure 8. The boundary of the outermost grid element of the model is fixed, and a certain displacement load is applied through the reference point of the rigid body of the fuel rod, and the direction is the surface where the vertical spring is located. In the grid stiffness analysis of different plan, except for the different models, the other constraints and loading conditions are the same. In order to facilitate finite element analysis, the position of welds, springs, etc. of the model is appropriately simplified during modeling, which will not affect the finite element analysis results. In addition, since the rigid-convex stiffness is much greater than the spring stiffness, the influence of rigid-convex deformation is ignored in the analysis, and it is considered that after the fuel rod is inserted into the corner grid element, the deformation is completely borne by the spring in the grid element.

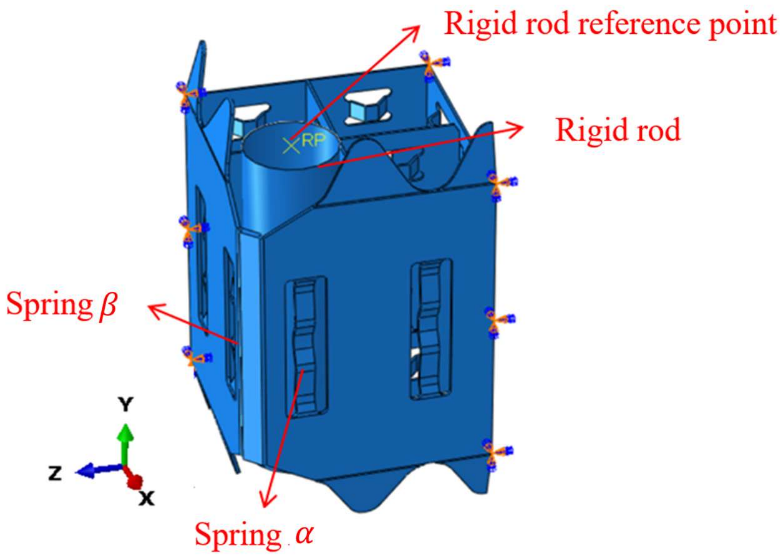


Figure 9 The assemble figure of element stiffness analysis

Parameters

The material used in the analysis was GH4169 alloy (Inconel 718), and the physical parameters of the material used in the analysis were entered according to the Fuel Material Performance Manual, as shown in Table 1. The displacements loaded in the analysis are determined by the analysis based on the actual dimensions of the model. In addition, the damage of the material is considered in the lattice hooking analysis, and the GH4169 alloy damage model adopts the ductile metal damage criterion, and the relevant damage parameters are determined with reference to the value method in the ABAQUA help document.

Table 1: Material properties

Name	Parameters	Value	Units
GH4169	Elastic modulus	180	GPa
	Poisson's ratio	0.3	-
	Yield strength	1034	MPa
	Tensile strength	1310	MPa

Name	Parameters	Value	Units
	Maximum tensile strain	0.12	-
	Density	6.25×103	kg/m3

Table 2: Analysis parameters

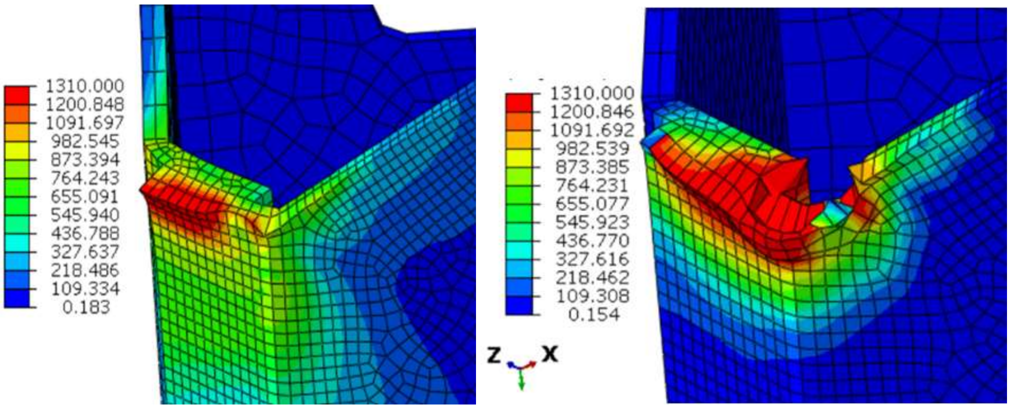
Type	Parameters	Value	Units
Hooking analysis	Displacement loading of movable fuel assmbly	4	mm
	Damage initiation strain	0.12	-
	Johnson-Cook damage d_1	0.11	mm
	Johnson-Cook damage d_2	0.75	-
	Johnson-Cook damage d_3	-1.45	-
	Johnson-Cook damage d_4	0.04	-
	Johnson-Cook damage d_5	0.89	-
	Sliding friction coefficient	0.28	-
Lattice stiffness analysis	Displacement loading of rod	1	mm

RESULTS

Hooking analysis

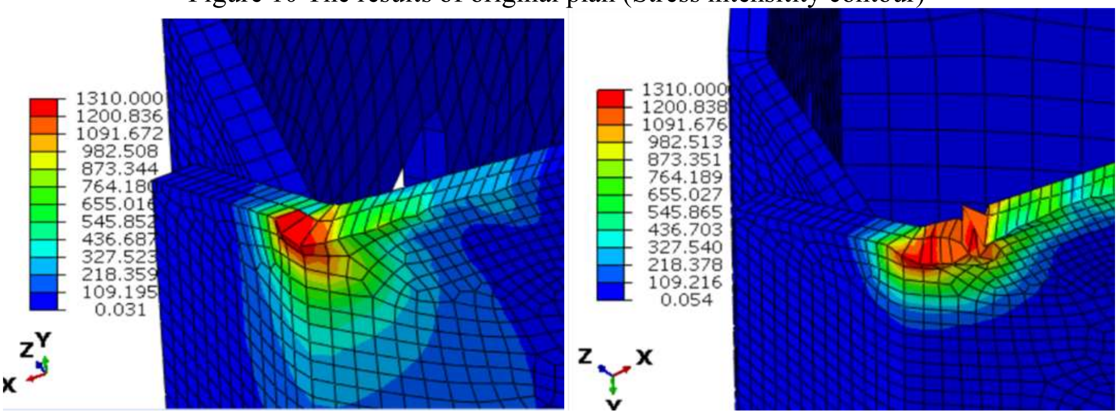
The hooking process of the lattice of different plan is simulated and analyzed, and the equivalent stress results of the hooking time of the fixed grid element and the movable grid element angle position of the lattice of different plan are extracted, as shown in Figure 10 to Figure 12. It is found that the angle grid element of the unimproved plan has obvious hooking phenomenon, and the angle position of the movable grid element is seriously scratched. After the improvement of plan A, there is no obvious hooking phenomenon in the lattice frame, but due to the influence of the bending process of the grid element strip, there is still a small platform at the bending place, resulting in the scratching phenomenon of a small area at the platform position, but no obvious damage occurs in the scratching area, indicating that the improvement of plan A can avoid the occurrence of hooking to a certain extent.For plan B, the scratching also occurs at the bending platform, and from the calculation results, compared with plan A, the area and deformation of the scraping area of plan B are smaller, and compared with the equivalent stress contour diagram, the grid angle position of plan B at the scraping time also has the smallest stress-affected area.

In summary, the two improved plan can effectively inhibit the occurrence of the phenomenon of hooking at the corner of the lattice frame, and due to the influence of the bending process, there is a small platform in the bending place of the improved lattice frame, which leads to the occurrence of hanging friction at the platform during the simulation process, and it is recommended to treat the platform area to avoid the scratching phenomenon. From the perspective of the stress-affected area, the anti-hanging effect of plan B is more superior.



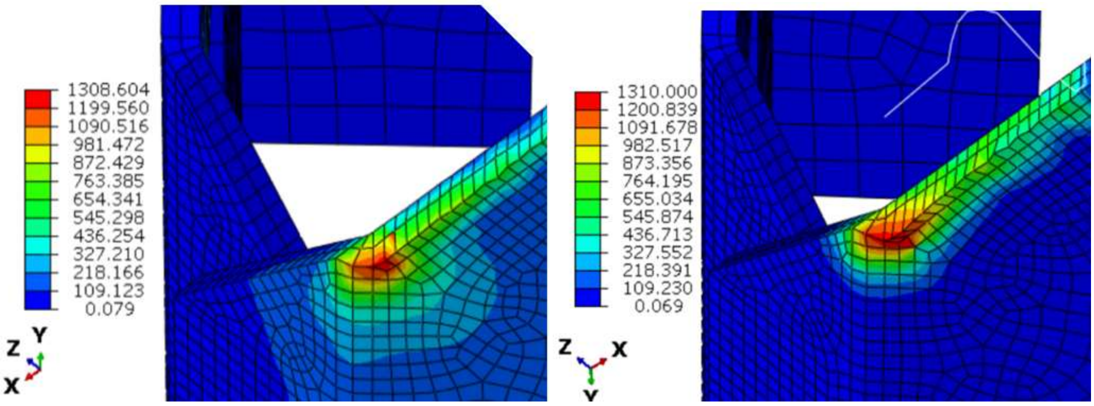
a) Fixed element b) Movable element

Figure 10 The results of original plan (Stress intensity contour)



a) Fixed element b) Movable element

Figure 11 The results of optimized plan A (Stress intensity contour)



a) Fixed element b) Movable element

Figure 12 The results of optimized plan B (Stress intensity contour)

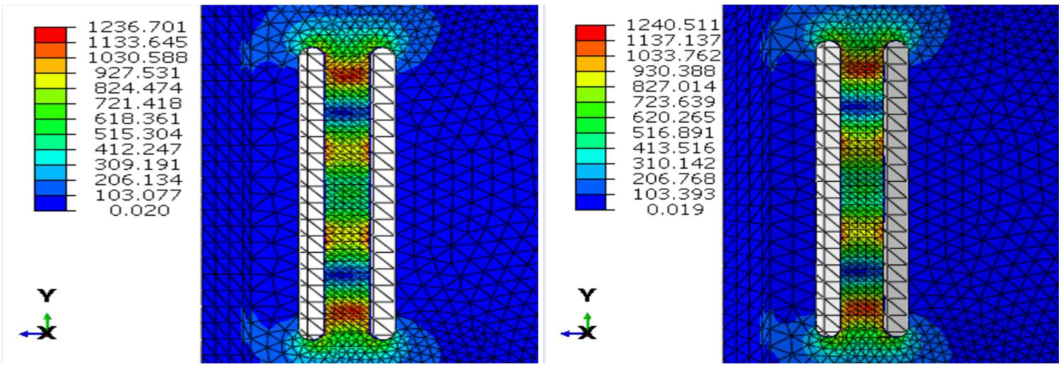
Lattice stiffness analysis

The spring stiffness of the lattice angle grid element of different plan is calculated and analyzed, and the spring stiffness is calculated by taking the ratio of the corresponding load to the deformation under the maximum deformation of the spring, wherein the spring deformation is the nominal value of the fuel rod diameter minus the nominal value of the stiff convex and spring spacing, and the equivalent stress of 0.65mm corresponding to the loading of 0.65mm is shown in Figure 13 and Figure 14. The load-displacement curves for extracting the fuel rods from the reference point are shown in Figures 15 and 16. It is found that the load displacement curve of the improved lattice is lower than that of the unimproved lattice, indicating that the chamfering angle will reduce the grid stiffness of the lattice, and the more material is removed, the more obvious the stiffness decreases. Table 3 shows the calculation results of grid element stiffness and grid element clamping force, the maximum reduction of grid stiffness in plan A is 0.34%, and the maximum reduction of grid element clamping force is 0.33%. The maximum reduction of the gate stiffness of plan B is 1.12%, and the maximum reduction of the gate element stiffness is 1.19%. The formula for calculating the stiffness of the gate element is:

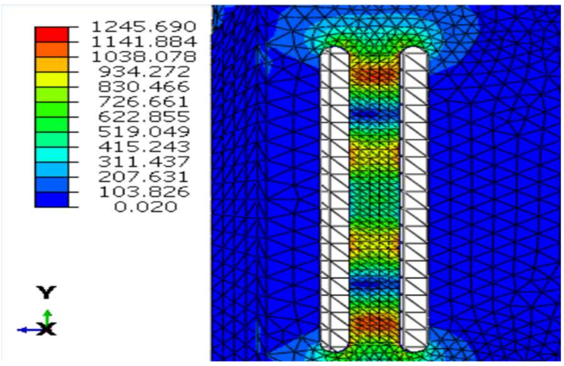
$$K = \frac{F}{X} \tag{1}$$

Where F is the clamping force of the grid element, unit N; K is the stiffness of the grid element, unit: N/mm; X is the deformation of the gate element, unit, mm. The change of the clamping force of the grid element caused by the dimensional tolerance is 0.235mm/0.65mm=36.1%, which is much greater than the change of the clamping force of the grid element caused by the improvement of the scheme (the maximum is 1.19%). The diameter of the fuel rod is 10±0.035mm, and the maximum size tolerance of the two is 0.235mm. The change of the clamping force of the grid element caused by the dimensional tolerance is 0.235mm/0.65mm=36.1%, which is much greater than the change of the clamping force of the grid element caused by the improvement of the plan (maximum 1.19%). In addition, the spring stiffness of the grid element is close to the change of the clamping force of the grid element, so the improvement of the two plan has little influence on the clamping force and spring rate of the grid element.

To sum up, there is no significant change in the clamping force and spring stiffness of the two improved plan. That is, the influence of the improved lattice plan on the blessing force and spring stiffness of the grid element is acceptable.

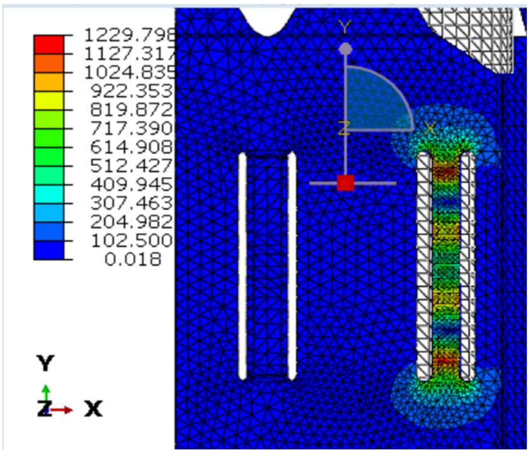


a) Original plan b) Plan A

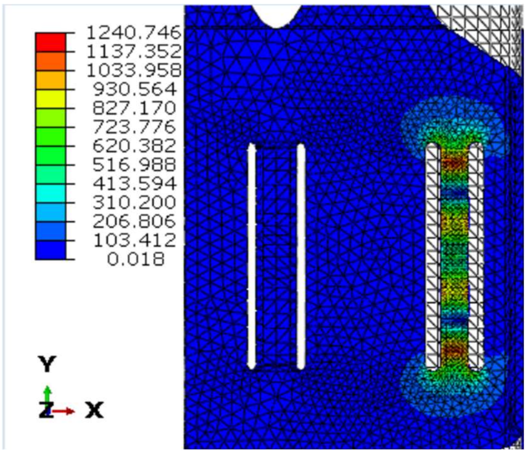


c) Plan B

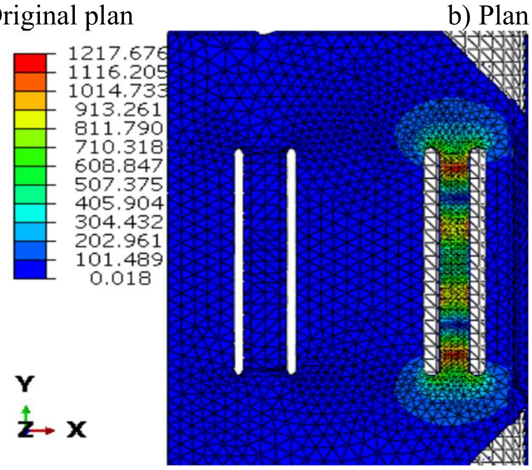
Figure 13. Calculation results of spring α for different plan



a) Original plan



b) Plan A



c) Plan B

Figure 14. Calculation results of spring β for different plan

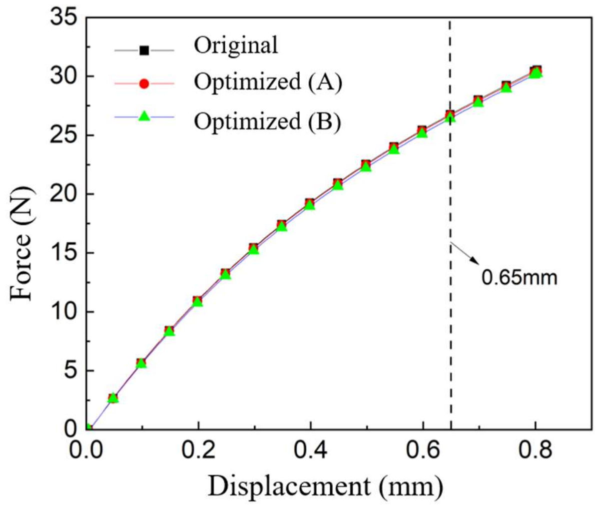


图 15 Force-displacement curves (Spring α)

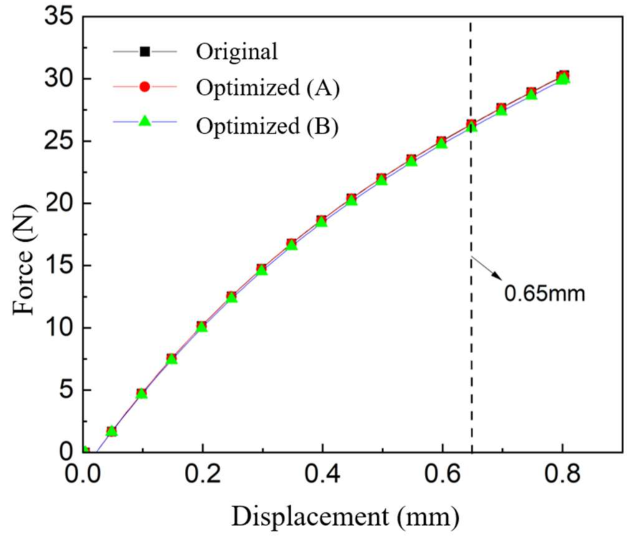


Figure16. Force-displacement curves (Spring β)

Table 3: The results of different lattice stiffness analysis

Spring Type	Lattice Type	Lattice clamping force at 0.65mm	Relative differences	Lattice spring stiffness at 0.65mm	Relative differences
Spring α	Original plan	27N	-	41.54 N/mm	-
	Plan A	26.91N	0.33%	41.40 N/mm	0.34%
	Plan B	26.68N	1.19%	41.05 N/mm	1.12%
	Original	26.89N	-	41.37 N/mm	-

Spring β	plan				
	Plan A	26.86N	0.11%	41.32 N/mm	0.12%
	Plan B	26.60N	1.08%	40.92 N/mm	1.09%

CONCLUSION

1. According to the design improvement scheme of the fuel assembly lattice, the grid hooking simulation analysis was carried out. The deformation of the grid corner lattice during the hooking hanging of the lattice angle is obtained. The analysis results are similar to the test results, and the anti-hooking performance of the lattice can be verified to a certain extent.

2. The anti-hooking performance of the two improved lattice schemes is significantly improved compared with that of the unimproved lattice scheme. Due to the influence of the bending process, there is a small platform at the bend of the corner lattice in the improved lattice frame, which leads to slight deformation at this position during the simulation process, and it is recommended to treat the bending area appropriately to improve the scratching phenomenon. From the perspective of the stress-affected area, the anti-hanging effect of plan B is more effective.

3. A finite element model for the calculation of the stiffness of the grid element was established, and the spring stiffness of the affected outer strip was analyzed. The results show that the improvement of plan A and plan B makes the gate element stiffness slightly reduced. After the improvement of scheme A, the stiffness of the gate element is reduced by 0.34% and the clamping force of the gate element is reduced by 0.33%. After the improvement of scheme B, the stiffness of the grid element is reduced by 1.12% and the clamping force of the grid element is reduced by 1.19%. The change of grid clamping force caused by the structural change of the lattice of the two schemes is much smaller than the change that may be caused by the manufacturing tolerance. Therefore, the influence of the improvement scheme on the original design scheme is negligible

In this study, the finite element method is used to simulate the process of lattice hook test, and the analysis results can provide some guidance and basis for the design improvement scheme of fuel assembly. This allows the effectiveness of the improvement program to be analyzed prior to manufacturing, which helps to shorten the design improvement cycle of the grid fuel assembly rack.