Transactions of the SMiRT 27
Yokohama, Japan, March 3-8, 2024
Division III

THERMAL STRESS DISTRIBUTIONS OF HEATPIPE COOLED REACTOR CORES: A REPRESENTATIVE VOLUME ELEMENT (RVE) MODEL STUDY

Yaqi Lin[1], Ding Zhou[2], Jiacheng luo[2], Lei Sun[2], Xiao Yan[2], Pengzhou Li[2]

[1] Research Scientist, Nuclear Power Institute of China, Chengdu, China
(linyaqi100027@163.com)
[2] Research Scientist, Nuclear Power Institute of China, Chengdu, China

ABSTRACT

With the continuous improvement of the demand for space technology, energy and power issues have gradually become a bottleneck in improving the performance of space equipment. As one of the preferred reactor types for the application of micro-unmanned nuclear power in the space field, the heatpipe reactor has strong development potential prospect. In the preliminary design of the heatpipe reactor core, the analysis of the high-temperature mechanical properties of the core matrix is a crucial step. Due to the large number of openings in the matrix and its strong periodicity, analysis is often performed on a single matrix module. The commonly used adiabatic boundary conditions are inconsistent with the actual situation of the matrix module, and cannot accurately simulate the interaction between the matrix modules on the target matrix. To address this problem, this paper proposes a matrix thermal stress representative volume element calculation method. First, based on the geometric symmetry and physical symmetry of the matrix structure, the feasibility of applying periodic boundary conditions (PBC) to the matrix module was analyzed. Then, the ANSYS program was used to establish coupling equations for the corresponding nodes on the corresponding boundary surfaces of the matrix module. The thermal stress of the matrix is calculated using periodic boundary conditions. Finally, three boundary condition combinations of different thermal boundaries and displacement boundaries are applied to compare with the calculation results of the full model of the matrix. The results prove the effectiveness of the calculation method proposed in this article, and can provide a more accurate method for the preliminary design process of the heatpipe reactor core.

INTRODUCTION

With the continuous improvement of space technology capacity demand, energy power problem has gradually become a bottleneck for further improvement of the performance of various space equipment, and needs to be solved urgently. Long endurance and intelligent control of energy power technology is an important basic technology related to the development of national space resources in the future and the maintenance of national security [1]. Compared with conventional energy, reactor power has many advantages such as high energy density, long life, small volume, high mobility and strong environmental adaptability, which can fundamentally solve the power shortcomings of space equipment and is the best choice for special energy in deep space and other fields in the future [2]. Among many candidate reactor types, heat pipe cooled reactor (referred to as heat pipe reactor) is essentially different from traditional loop reactor. Instead of driving fluid to bring out the core fission energy through pump valves, this reactor uses a large number of independent alkali metal high-temperature heat pipes to passively export the core fission energy. Heat pipe reactor is considered to be one of the preferred reactor types for the application of micro-small unmanned nuclear power in the space field [3].

Heat pipe reactor refers to a solid state reactor where the primary loop system does not adopt the coolant loop layout, but adopts heat pipes to conduct the heat generated by the core to the secondary loop system or thermoelectric conversion device [4]. When the heat pipe cooled reactor is running, the fission energy generated by the reactor is conducted to the evaporating section of the metal heat pipe arranged in the core. Through the evaporation and condensation process of the working medium inside the heat pipe and the natural circulation flow, the heat is conducted from the core to the hot end of the thermoelectric conversion device/secondary loop system. After the thermoelectric conversion device/secondary loop system converts the heat energy into electrical energy, the remaining waste heat is discharged to the final heat trap (atmosphere or environment) through coolers or radiative radiators [5]. The heat pipe cooled reactor system consists of the reactor body, high-temperature heat pipe, thermoelectric conversion device, waste heat discharge system, electricity load, standby load and reliable power supply.

For the bulk matrix of the core, Ma Yuguang et al. [7] conducted nuclear thermal coupling research on the core. The results showed that the SS-316 matrix could not withstand the peak pressure generated during the operation of the core, so Ma recommended the Mo-Re alloy with higher strength as the core matrix. In addition, since the Mo-14Re alloy has strong corrosion resistance and compressive strength and has been widely used in space heat pipe reactor research, Ma did not analyze the neutron science of the Mo-Re alloy matrix core. Since SiC material has good corrosion resistance, high compressive strength and good neutron economy and has been widely used in new reactor fuel matrix or cladding material, Hernandez recommended the use of SiC instead of SS-316 matrix in heat pipe reactor [8].

In the preliminary design of the matrix structure, due to the large number of openings and the periodic characteristics of the module arrangement, the analysis is often focused on a single module as shown in Figure 1 Los Alamos Laboratory conducted a thermal stress study on two different structures (6pin and 4pin) of the matrix module in 2002[9]. Among them, the 6pin module is the Mars surface heat pipe reactor, and the 4pin module is the NEP heat pipe reactor. The thermal stress distribution calculation results are shown in Figure 2 and Figure 3. However, Los Alamos Laboratory used adiabatic boundary conditions in the thermal steady-state calculation, which is inconsistent with the actual thermal boundary conditions of the matrix module. In the complete matrix, the matrix module is surrounded by other matrix modules, and the matrix modules are arranged compactly and affect each other. This paper aims to propose a more realistic thermal stress calculation method, which can include the influence of the surrounding matrix modules on the intermediate matrix module, which will improve the accuracy of the calculation in the preliminary design stage of the matrix structure.

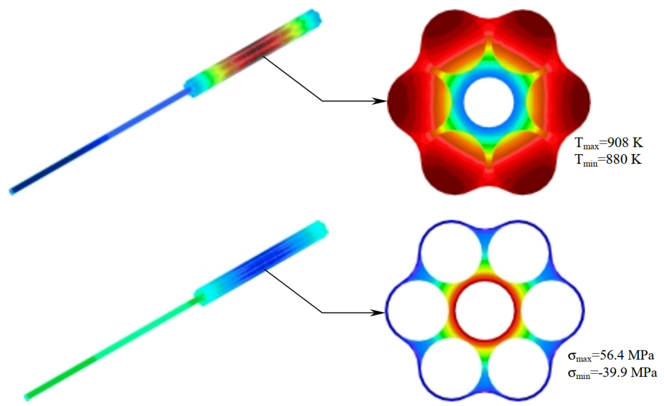
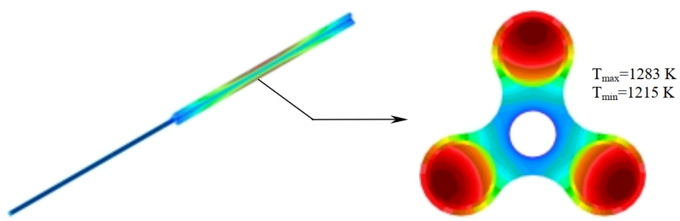


Figure 1. Temperature distribution and thermal stress calculation results of 6pin module



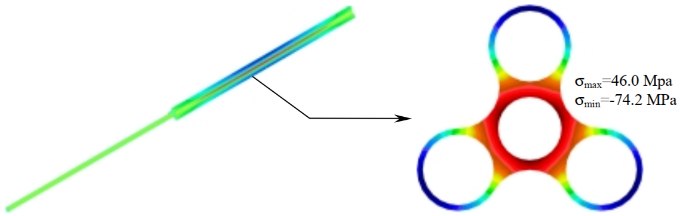


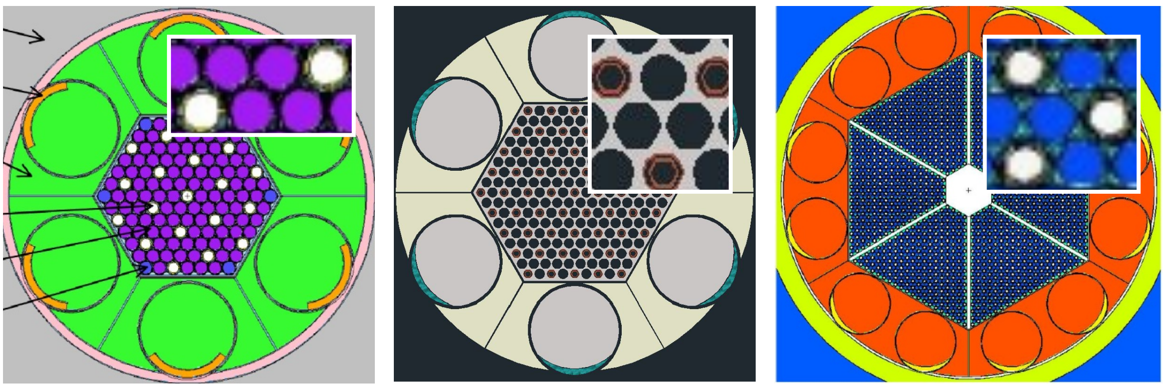
Figure 2. Temperature distribution and thermal stress calculation results of 3pin module

MATRIX STRUCTURE SYMMETRY ANALYSIS

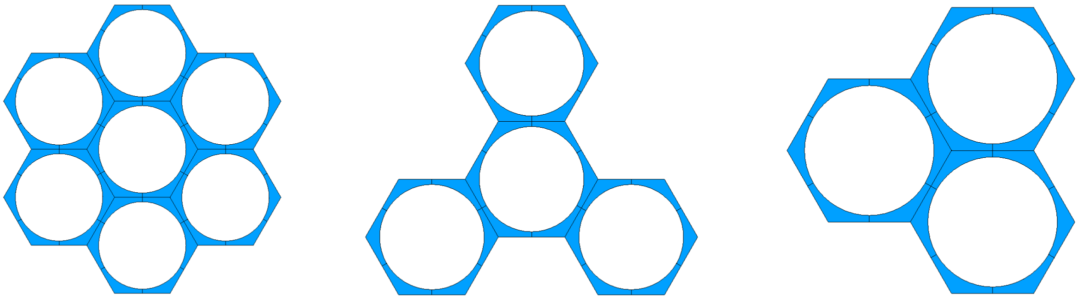
Matrix is a structure with regular repeating characteristics, and its symmetry is reflected in two aspects: geometric symmetry and physical symmetry. For geometric symmetry, the separation and selection of matrix modules are mainly studied. For physical symmetry, the boundary conditions of matrix modules are mainly studied. In mechanical analysis, the common symmetry is mainly axial symmetry and cyclic symmetry, and the common cyclic symmetry can be divided into rotational symmetry and translational symmetry. The former can correspond to the blade of the engine, and in numerical analysis, a single blade is usually modeled and rotated period boundary conditions are applied. The latter is common in the micro-scale mechanical performance analysis of multi-scale mechanical analysis of composite materials, and representative volume element (RVE) is generally used to construct the micro-scale structure and performance, and periodic boundary conditions (PBC) are usually applied to it.

GEOMETRIC SYMMETRY

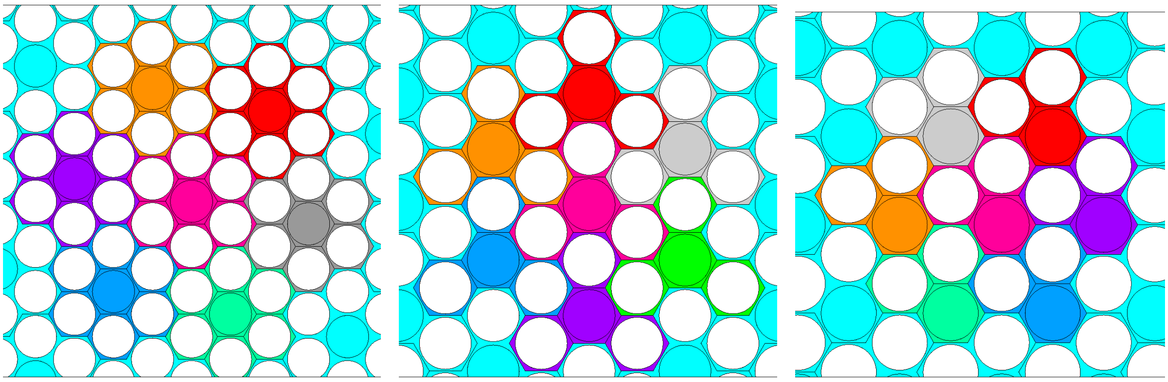
There are three typical matrix heat pipe/fuel rod arrangements. According to the number of heat pipes in a single matrix module, they can be divided into 6pin, 3pin, and 2pin structures, as shown in Figure 3. These three structures can be distinguished according to the number and arrangement of fuel rods between the heat pipes of adjacent matrix modules. The 6pin, 3pin, and 2pin translationally symmetric units are extracted respectively, as shown in Figure 5. The representative unit in Figure 5 is translated symmetrically to obtain Figure 6, where the three representative units can all complete dense laying. Therefore, these three matrix modules are parallel symmetric units, and the geometry symmetry of the matrix is proved.



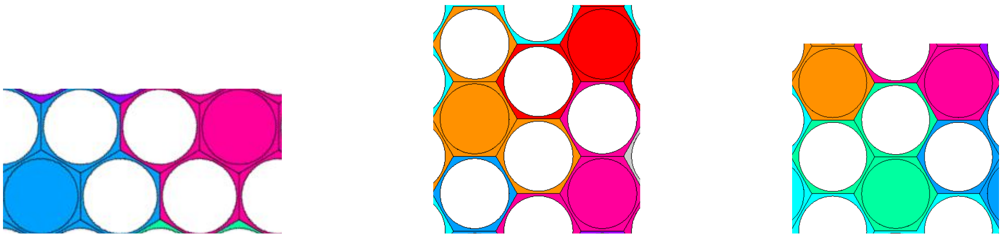
(a) 6pin structure [3] (b) 3pin structure [4] (c) 2pin structure [4]
Figure 3. 6pin, 3pin, and 2pin core structure design drawings



(a) 6pin represents the body unit (b) 3pin represents the body unit (c) 2pin represents the body unit
Figure 4. 6pin, 3pin, and 2pin RVE core structure design



(a)6pin represents the body unit (b) 3pin represents the body unit (c) 2pin represents the body unit
Figure 5. 6pin, 3pin, and 2pin RVE core structure design



(a) 6pin represents the body unit (b) 3pin represents the body unit (c) 2pin represents the body unit
Figure 6. 6pin, 3pin, and 2pin RVE core local part

Comparing the local enlarged images of Figure 4 with Figure 6, it can be seen that the local characteristics are consistent. Therefore, the unit in Figure 5 meets the condition of geometric symmetry and can be used as the matrix module of the three typical heat pipe reactor core structures.

PHYSICAL SYMMETRY

Periodic Boundary Conditions

Due to the rapid development of numerical simulation technology, people have realized the simulation of real physical systems through numerical calculation. Although the running speed of computers has been greatly improved, the computing power of PCs is still beyond the large-scale macro calculation. Assuming that materials with meso-structure are statistically uniform, the meso-information of materials can be obtained by infinite copying a window, which is called a representative volume element. First of all, the size of the representative element is much smaller than the size of the macro structure. In addition, the representative element needs to be representative of the meso-structure information, and the superposition in any direction can ensure coherence without displacement. Therefore, in the finite element simulation, composite materials with meso-structure can be seen as periodic arrangements of multiple representative volume elements (RVE). When periodic arrangements are carried out, in order to ensure geometric compatibility, periodic boundary conditions must be met [10].

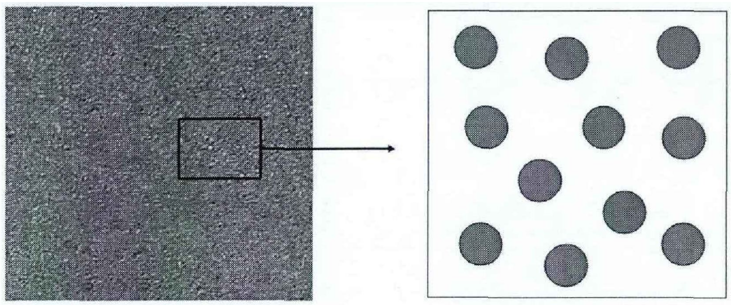


Figure 7. Two-dimensional RVE representative element of mesomechanics

In order to ensure the deformation coordination of adjacent representative units, the concept of periodic boundary is introduced. The representative units with periodic boundary have truly realized the purpose of using meso-structure information to statistically analyze macro-properties, and are widely used in the fields of mesomechanics and molecular dynamics. In the implementation of finite element, the periodic boundary conditions are mainly realized through the following three ways: (1) the introduction of penalty function, which is easier to implement, but will cause numerical problems and errors;(2) based on the plane assumption, the implementation is achieved by imposing normal constraints and normal coupling, but due to the ideal boundary conditions, the formation of excessive constraints;(3) adding coupling equations, this method reduces the strength of constraints and improves the accuracy of simulation results, but due to the number of nodes, there will be some difficulties in numbering nodes. In order to meet the deformation coordination and achieve the purpose of meso-structure representing macro-structure, most numerical simulations adopt the plane assumption boundary conditions in the calculation process, that is, the second method of periodic boundary. According to the homogenization theory, the macroscopic mechanical properties of materials can be reflected by the average mechanical properties of representative volume units. Although the second type of periodic boundary satisfies the requirements of deformation coordination, it overly constrains the boundary, destroys the periodicity of boundary stress, and does not meet the conditions of infinite superposition. Periodic representative units not only satisfy the deformation coordination on the boundary, but also ensure the continuity of stress. On the boundary surface of the representative unit, the displacement of the strictly periodic boundary conditions can be expressed as [11]:

$$u_i = \bar{\varepsilon}_{ik} x_k + u_i^* \tag{1}$$

In the equation, $\bar{\varepsilon}_{ik}$ is the average strain of the structure, representing a linearly distributed displacement field, u_i^* is the periodic displacement component of the boundary surface. Since the periodic arrangement of representative units can represent a continuous whole, the boundary of adjacent units needs to meet two continuous requirements. First, the displacement is continuous. After being deformed by force, the boundary of adjacent RVE cannot separate or overlap. In addition, the stress distribution of opposite boundaries of RVE should be the same. It is obvious that Equation (1) meets the requirement of displacement continuity. In the specific analysis process, the value of ui∗ needs to be determined. For any RVE, the opposite boundaries are paired. Taking the three-dimensional representative unit as an example, as shown in Figure 8, the displacement of node i at the corresponding position of the parallel boundary surface can be expressed as:

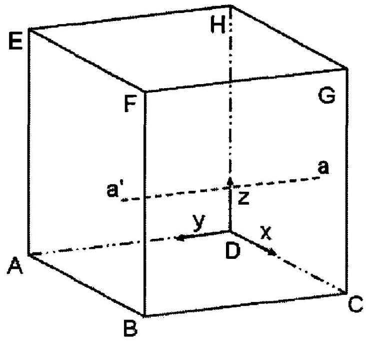


Figure 8. Cubic Unit

$$u_i^{a} = \bar{\varepsilon}_{ik} x_k^{a} + u_i^{*} \tag{2}$$

$$u_i^{a'} = \bar{\varepsilon}_{ik} x_k^{a'} + u_i^{*} \tag{3}$$

Subtract the two equations and you get:

$$u_i^{a'} - u_i^{a} = \bar{\varepsilon}_{ik}\left(x_k^{a'} - x_k^{a}\right) = \bar{\varepsilon}_{ik}\, \Delta\, x_k^{a} \tag{4}$$

Since the two opposite surfaces are parallel, and$\Delta\ x_k^{a}$ is a constant, Equation (4) can be written as:

$$u_i^{a'}(x, y, z) - u_i^{a}(x, y, z) = c^{n}\ (n = 1,2,3) \tag{5}$$

Constantc1 , c2 , c3 represent the average tensile or compressive forces in the x, y, and Z directions, respectively. Equation (5) realizes the periodicity and continuity of the upper boundary displacement, and the periodic boundary can be realized in the finite element analysis by applying the node coupling equation. However, if the displacement of the opposite boundary is not constant, Equation (5) cannot fully guarantee the continuous stress distribution of the boundary. The condition for stress continuity needs to be added [12]:

$$\sigma_i^{a'} - \sigma_i^{a} = 0,\ \tau_i^{a'} - \tau_i^{a} = 0 \tag{6}$$

In Equation (6), σ_i and τ_i respectively represent the normal stress and shear stress of the parallel boundary surface. Therefore, the combination of equations (5) and (6) can realize the addition of periodic boundary. Figure 9 is the deformation diagram of the two-dimensional model under the condition of periodic boundary. By imposing periodic boundary, the displacement difference of RVE units relative to the boundary is a constant, and a continuous whole can be obtained by representing the periodic arrangement of units.

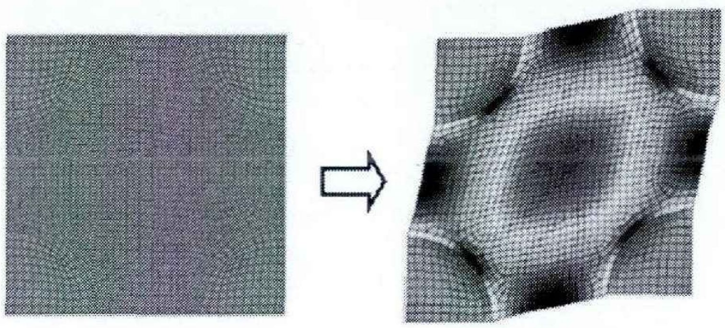


Figure 9. Two-dimensional deformation diagram under periodic boundary conditions

In commonly used commercial finite element analysis software, periodic boundary conditions are realized by adding coupling constraint equations on the basis of Equation (5). It is worth noting that the nodes adding coupling equations must be guaranteed to be on the opposite side, and the node positions must be one-to-one corresponding. Therefore, in the process of grid partition, it is necessary to ensure the consistent number of opposite side units and their corresponding positions.

Implementation Of Periodic Boundary Conditions In Finite Element Method

The representative units that satisfy the periodic boundary conditions have been established. This section mainly studies how to add periodic conditions in the finite element analysis process. Taking the three-dimensional representative unit as an example, the equation of the periodic boundary condition can be expressed as:

$$u_i^{a'} = u_i^{a} + u_i^{A}$$
$$u_i^{b'} = u_i^{b} + u_i^{B} \qquad (i = x, y, z) \qquad (7)$$
$$u_i^{c'} = u_i^{c} + u_i^{C}$$

The plane OAC, OAB, and OBC are the main surfaces, and the opposite surfaces are the secondary surfaces. The nodes on the main surfaces are the main nodes, and the nodes on the secondary surfaces are the secondary nodes. A, B, and C are the datum points, and the specific locations are shown in Figure 10.

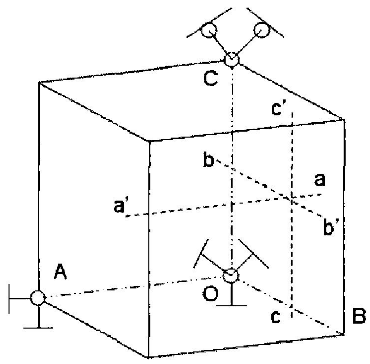
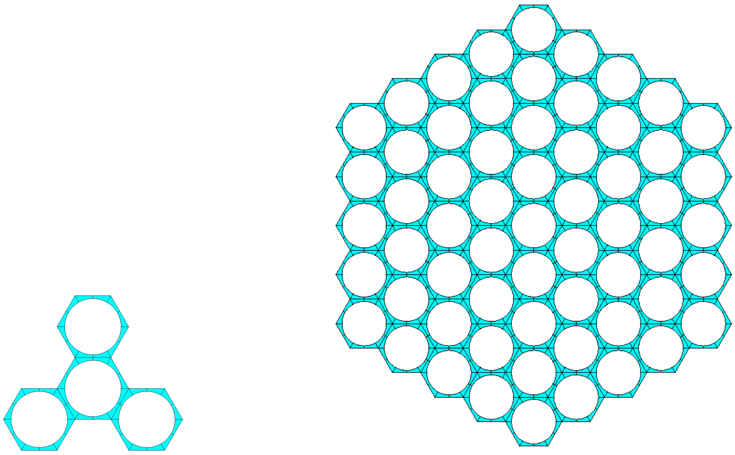


Figure 10. Schematic diagram of the periodic boundary of three-dimensional representative units

It is easy to establish coupling equations in ANSYS software, but due to the large number of nodes on the main surface and the secondary surface, it is time-consuming to establish coupling equations manually, and the selection of node pairs is easy to make mistakes. In order to save time and improve work efficiency, we can write APDL cycle commands to find the node position and number of the corresponding edge interface, and automatically generate coupling equations.

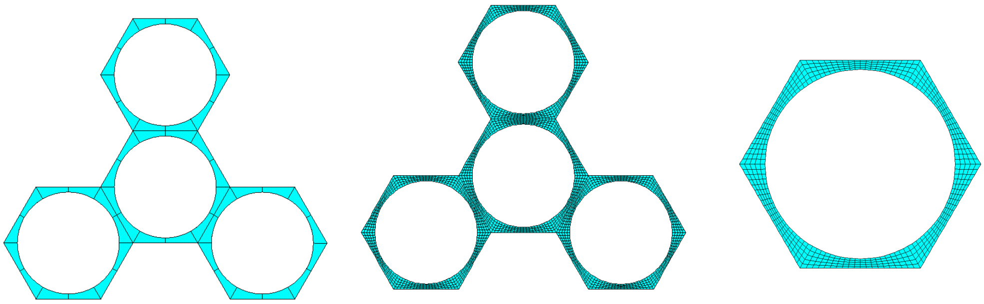
FINITE ELEMENT CALCULATION OF THERMAL STRESS IN MATRIX

The thermal expansion effect of the matrix will occur when the heat pipe reactor operates at high temperatures, which significantly affects the heat transfer and neutron physical transport process of the reactor. The matrix around the heat pipe will expand inward, making it difficult to extract the heat pipe and causing potential hazards to the safe operation of the reactor. Therefore, it is necessary to calculate the thermal stress of the matrix at the early stage of the design. Three boundary combinations are formed by imposing different thermal boundary and displacement boundary combinations on the matrix module, and the calculation results are compared with the full matrix model. In this chapter, the full matrix model and the matrix module model are established respectively. The thermal stress of the full model is calculated under the condition of free expansion, and the thermal stress of the matrix module model is calculated under the condition of thermal expansion and periodic boundary conditions, so as to evaluate the effectiveness and accuracy of the representative volume element method.



(a) Matrix module (b) Full matrix model

Figure 11. Geometrical models of different structural sizes



(a) Segmentation plane (b) Matrix module mesh. (c) Mesh section zoom

Figure 12. Matrix module mesh generation of finite element model

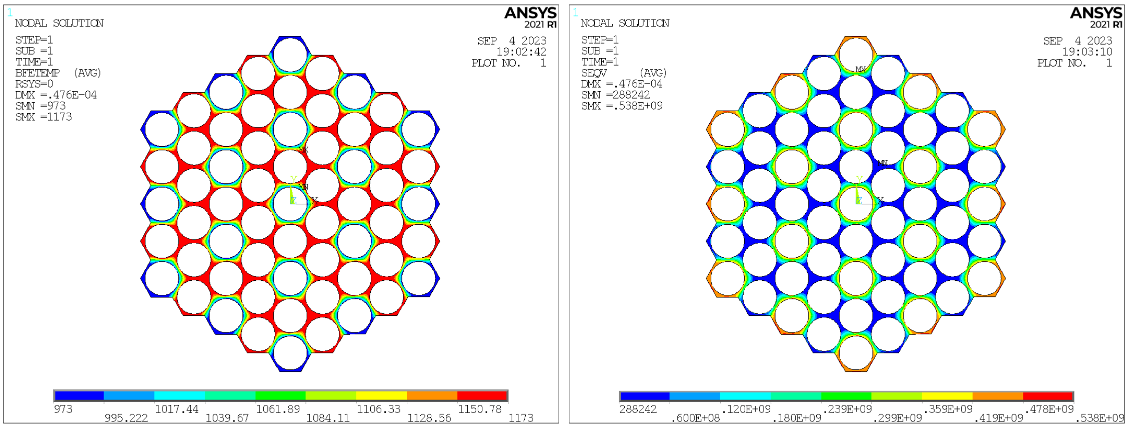
When establishing the two-dimensional model of the matrix module, the target surface is divided into multiple regular same surfaces by performing surface cutting on the two-dimensional model as shown in Figure 12(a). Then, by modifying the grid partitioning method and commanding MSHKEY to use mapping partitioning method on the target surface, the periodic grid as shown in Figure 12(b) can be generated. As can be seen from Figure 12(c), the number of opposite surface units is consistent and the positions are corresponding, which meets the requirements of imposing periodic boundary conditions.

CALCULATION RESULTS

This section mainly analyzes the influence of different boundary conditions on the calculation results. In the process of thermal stress calculation, the thermal steady-state analysis is carried out on the model first, and then the analysis results of thermal steady-state, namely the structure temperature distribution, are applied as temperature loads in the thermal stress calculation. For thermal steady-state analysis, the boundary conditions can be divided into two types: adiabatic boundary conditions and thermal periodic boundary conditions; for thermal stress analysis, the boundary conditions can be divided into free expansion boundary conditions and displacement periodic boundary conditions. The combination of boundary conditions, the calculation results of the four boundary combinations and the target difference with the control group are shown in Table 1.

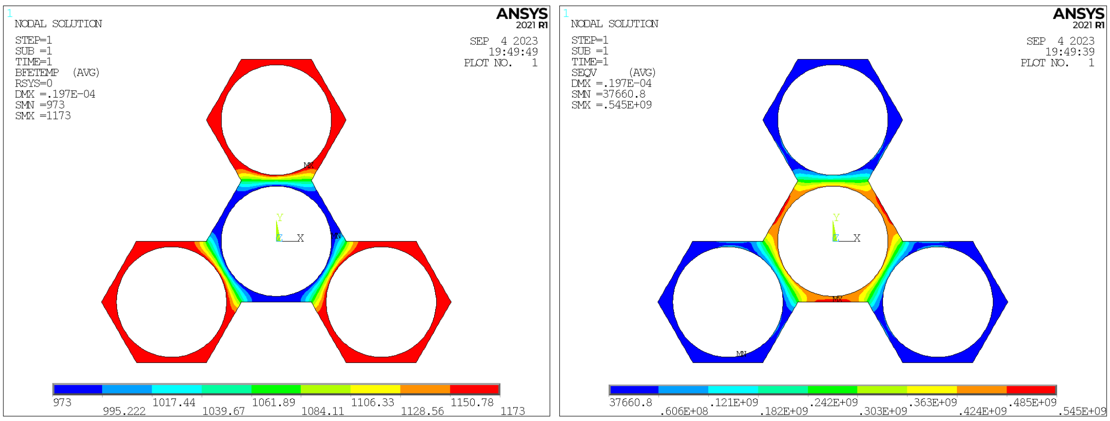
Table 1 Comparison of calculation results of different boundary conditions

Model Size	Boundary Condition Composition	Thermal Boundary	Displacement Boundary	Maximum Thermal Stress/MPa	Target Deviation
Full Model	1	Adiabatic	Free Expansion	500.5	
Matrix Element	2	Adiabatic	Free Expansion	545.2	8.9%
Matrix Element	3	PBC	Free Expansion	486.9	-2.7%
Matrix Element	4	PBC	PBC	503.6	0.6%



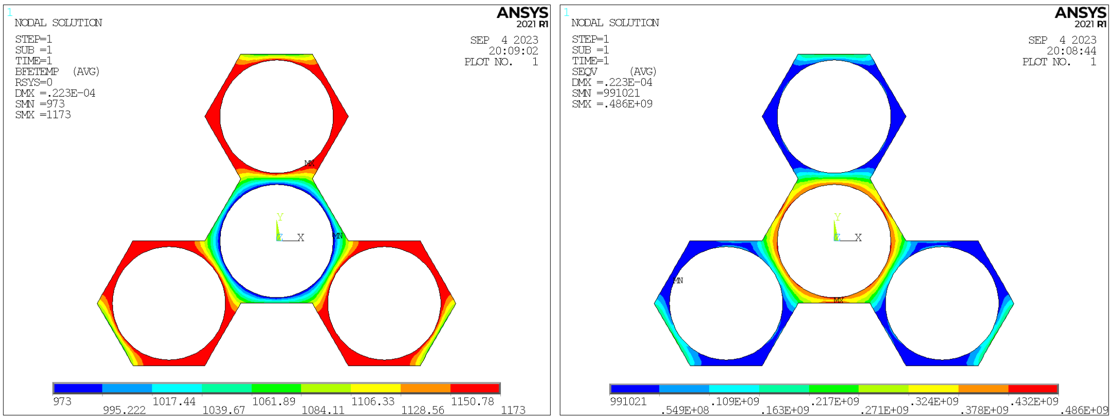
(a) Temperature distribution (b) Stress distribution

Figure 13. Calculation results of adiabatic free expansion for full model boundary combination 1



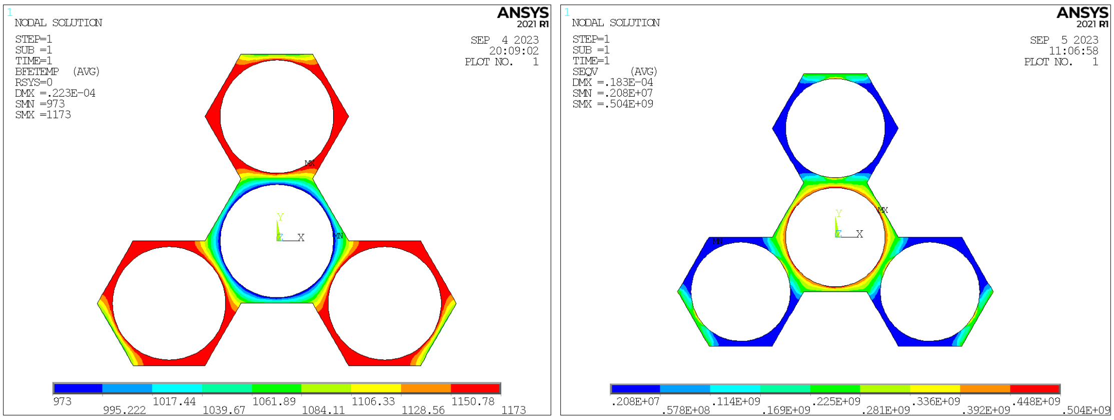
(a) Temperature distribution (b) Stress distribution

Figure 14. shows the adiabatic free expansion calculation results of boundary combination 2 of the representative volume element model



(a) Temperature distribution (b) Stress distribution

Figure 15. Thermal stress calculation results of free expansion of representative volume element model boundary combination 3 thermal PBC



(a) Temperature distribution (b) Stress distribution

Figure 16. Calculation results of thermal PBC displacement and thermal stress of representative volume element model boundary combination 4

The thermal stress distribution of the matrix between the heat pipe and the fuel rod under four boundary combinations was extracted, namely, path 1 and path 2 in Figure 17. The stress distribution curves were shown in Figure 18 and Figure 19. As can be seen from the curves, the stress change trend of boundary combination 1 and boundary combination 4 on path 1 and path 2 was better than that of the other two boundary combinations.

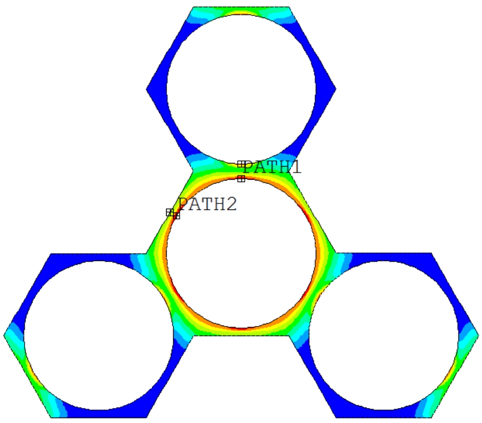


Figure 17. Extraction path

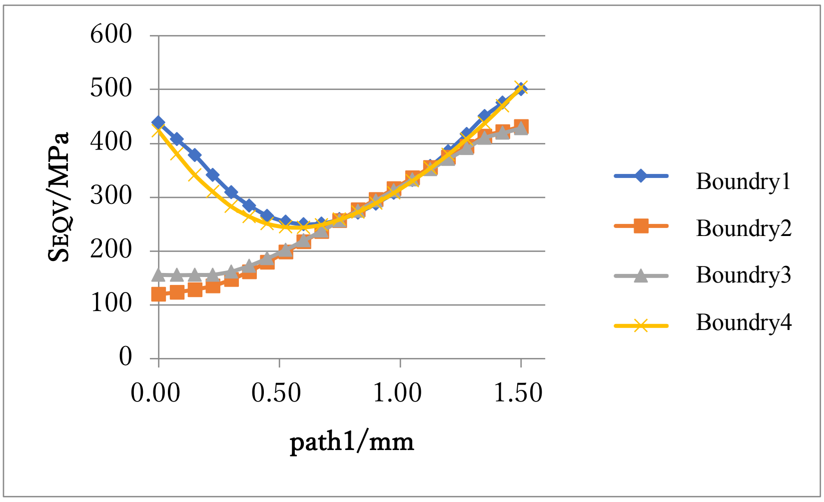


Figure 18. Stress variation trend along path 1

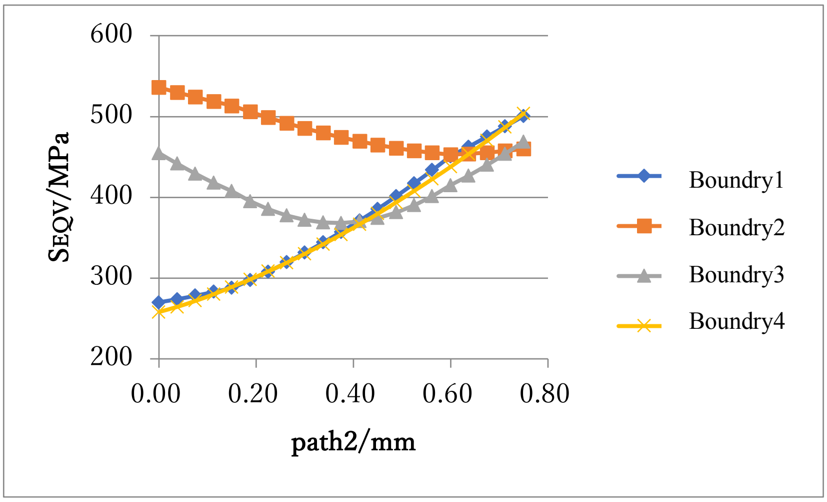


Figure 19. Stress variation trend along path 2

CONCLUSION

As one of the preferred reactors for micro-scale unmanned nuclear power in space applications, heat pipe reactor has a strong development prospect. In the preliminary design of the core of heat pipe reactor, the high-temperature mechanical characteristics of the core matrix are a very important step. Due to the large number of matrix openings and the strong periodicity of the matrix module arrangement, the analysis of a single matrix module is often carried out in the preliminary design of the matrix structure. In the thermal analysis of the matrix module, the commonly used adiabatic boundary conditions are inconsistent with the actual situation of the matrix module, and cannot accurately simulate the interaction between the matrix modules on the target matrix. In order to solve this problem, this paper proposes a calculation method for the matrix thermal stress representative volume element. Firstly, based on the geometric symmetry of the matrix structure, the matrix modules of three common heat pipe reactor core matrix structures are analyzed. Then, according to the physical symmetry, combined with the similarities between the matrix module and the representative volume element (RVE), the feasibility of applying Periodic Boundary Condition (PBC) on the matrix module is analyzed. Then, the APDL command is written by ANSYS program to establish the coupling equation on the corresponding node on the corresponding boundary surface of the matrix module, and the periodic boundary condition is applied. By imposing different combinations of thermal boundary and displacement boundary on the matrix module, the calculation results are compared with the calculation results of the whole matrix model. The calculation results show that the calculation results of the matrix thermal stress representative volume element method have the highest coincidence with the results of the free expansion of the whole matrix model from the two aspects of surface stress value and path stress change trend. This proves the effectiveness of the calculation method proposed in this paper, which can provide a more accurate method for the preliminary design process of the heat pipe reactor core.

REFERENCES

The development of heat pipe cooled reactors, Yu Hongxing, Ma Yugao, Zhang Zhuohua, Chai Xiaoming, Nuclear Power Engineering (2019)

Zhang Ming, Cai Xiaodong, Du Qing, et al. Research on space application of nuclear reactors [J]. Spacecraft Engineering, 2013, 22(6)

Hu Gu, Zhao Shouzhi. Overview of space nuclear reactor power supply technology [J]. Journal of Deep Space Exploration,2017(5)

Wu Haosong, Dai Ding, Zhang Shufeng, Li Jiaze. The United States actively promotes the development and deployment of space nuclear power systems [J]. Foreign Nuclear News, 2021(01).

KILOPOWER: NASA'S OFFWORLD NUCLEAR REACTOR.

Joshua C Walter, Thermal Expansion and Reactivity Feedback analysis for the HOMER-15 and SAFE-300 Reactors.

Yugao Ma, Research on Nuclear Thermal-mechanical Coupling of Heat Tube Cooled Reactor [J]. Nuclear Power Engineering, 2020(4)

Patrick R Mcclure, Design of Megawatt Power Level Heat Pipe Reactor.

Thermal Stress Calculations for Heatpipe-Cooled Reactor Power Systems, Los Alamos National Laboratory.

Chen Zhe, Experimental Study and Finite Element Simulation of Monotonic and Cyclic Deformation Behavior of Aluminum Foam [D]. Southwest Jiaotong University, 2013.

Tyrus J.M., A Local Finite Element Implementation for Imposing Periodic Boundary Conditions on Composite Micromechanical Models. Solids and Structures. 2007, 44.

Hollister S.J.,A comparison of homogenization and standard mechanics analysis for periodic porous composites. Computational Mechanics. 1992, 10.