Transactions of the Korean Nuclear Society Autumn Meeting
Pyeongchang, Korea, October 30-31, 2014

Structural Analysis of the Upper Internal Structure in PGSFR

S. H. Kim [a*], G. H. Koo [a]
[a]*Korea Atomic Energy Research Institute, Daejeon 305-600, The Republic of Korea*
[*]*Corresponding author: shkim5@kaeri.re.kr*

1. Introduction

The upper internal structure (UIS) is a package of hardware suspended from the rotating plug to about 20 cm above the core assemblies. The functions of the UIS are to support shroud tubes containing the primary and secondary control rod drivelines and preserve critical alignments between these drivelines and the core lattice, under normal and off-normal conditions. In addition, the UIS produces sufficient coolant mixing to mitigate thermal transients to downstream components and provides an opening for the In-Vessel transfer machine to access inner core positions without interfacing with the control rod drive lines and the upper core instrumentation package [1]. Fig. 1 shows the current UIS design in the PGSFR.

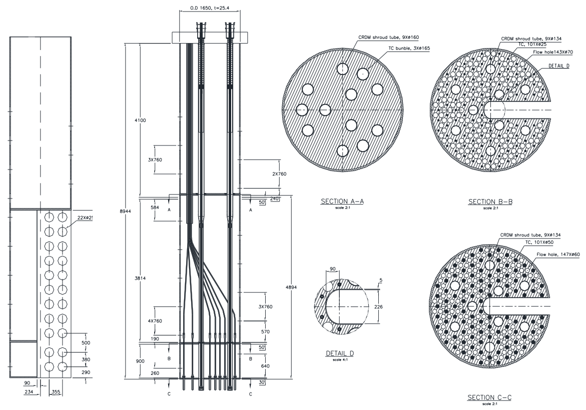


Fig.1 Design configuration of the UIS

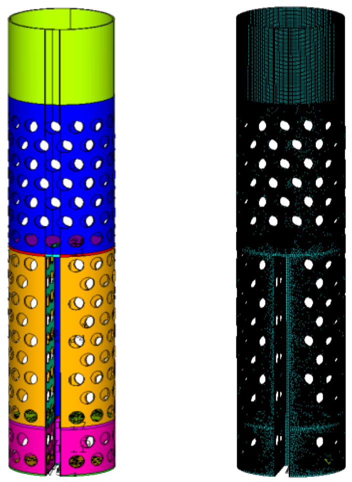


Fig.2 Analysis model of the UIS

The outer boundary is a cylindrical tube type, which is perforated for the release of coolant to the hot pool at an elevation close to the free surface. The radial position of the shroud tube is fixed by three horizontal guide plates and the lower guide plate is close to the core assemblies and is perforated to permit most of the core effluent to reach the region between guide plates. The material of the UIS is 316 stainless steel and the analysis is performed on the basis of the 3D solid model as shown in Fig. 2.

In this study, the primary stress analysis for dead weight was carried out and the thermal stress analysis considering the coolant temperature around the UIS was performed. In addition, the mode characteristics of the structure by the natural frequency analysis were evaluated.

2. Structural Analysis for the UIS

2.1 Primary Stress Analysis for Dead Weight

The overall length of the UIS is 8.944 m and its outer diameter is 1.65 m. The control rod assemblies and instrumentation drywells installed in the inside of the UIS is not considered in the analysis because they are separately supported by the rotating plug. The primary stress analysis is carried out for the dead weight. Fig. 3 and Fig. 4 represent the stress intensity distribution due to the dead weight. As shown in Fig. 4, the maximum stress intensity is 2.39 MPa, which occurs at the local discontinuity of the upper part of the slot. Fig. 5 shows the displacement distribution of the UIS structure. From this figure, we can see that the maximum displacement of 0.4 mm occurs at the bottom side.

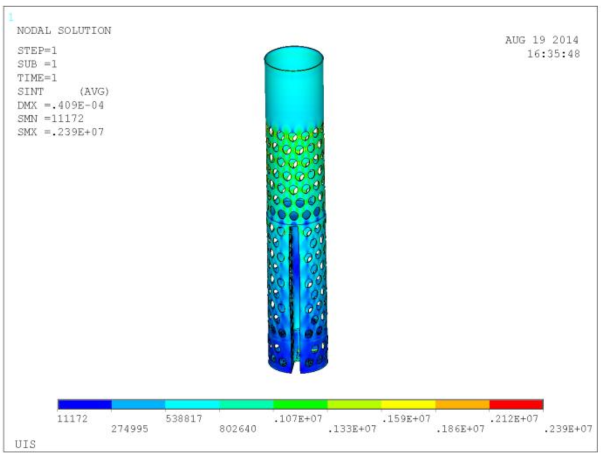


Fig.3 Stress intensity distribution for the dead weight

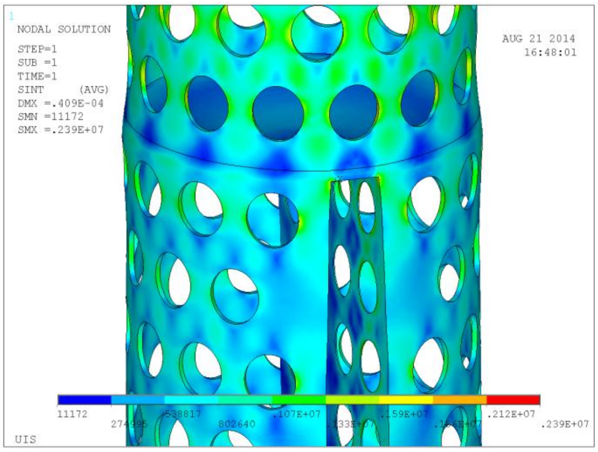


Fig.4 Maximum stress intensity for the dead weight

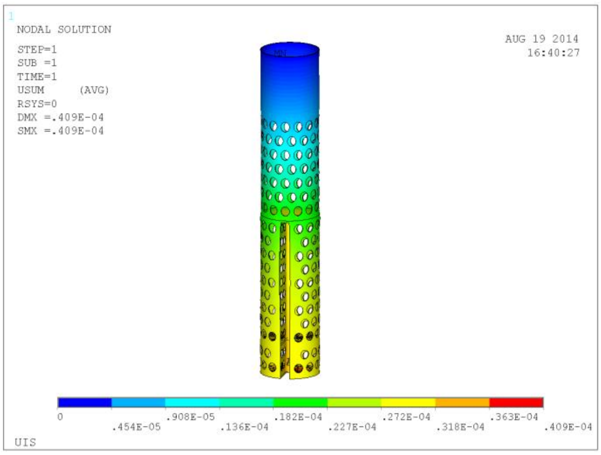


Fig.5 Displacement for the dead weight

2.2. Thermal Stress Analysis

In the thermal stress analysis, the coolant temperature around the UIS and the heat convection coefficient are considered as the thermal loads. As a structural boundary condition, the top surface of the UIS is constrained in the axial and circumferential direction. The ambient coolant temperature is 545 ^{o}C and the heat convection coefficients in the sodium region and the cover gas region are 10000 W/^{o}C-m^{2} and 2.2783 W/^{o}C-m^{2}, respectively.

As a result of the analysis, the temperature distribution of the UIS due to the thermal load is calculated as shown in Fig. 6. The maximum temperature is 545 ^{o}C, which occurs in the cylindrical part immersed in the hot sodium pool. Fig. 7 and Fig. 8 show the stress intensity distribution for the UIS. As shown in these figures, the maximum stress intensity is 38.7 MPa, which is generated at the thermal transition area in the upper part of the UIS. In addition, the maximum displacement as shown in Fig. 9 is calculated as 8 cm.

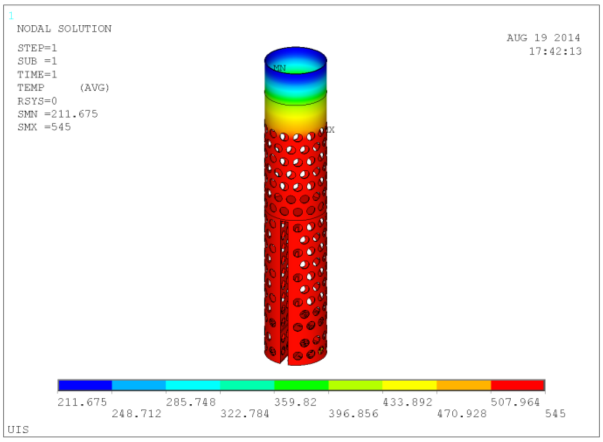


Fig. 6 Temperature distribution for the thermal load

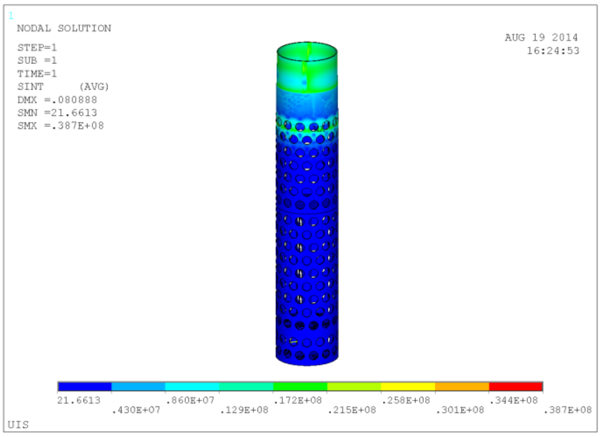


Fig. 7 Stress intensity distribution for the thermal load

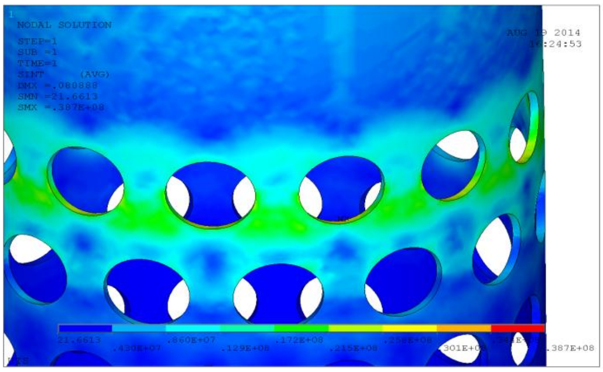


Fig. 8 Maximum stress intensity for the thermal load

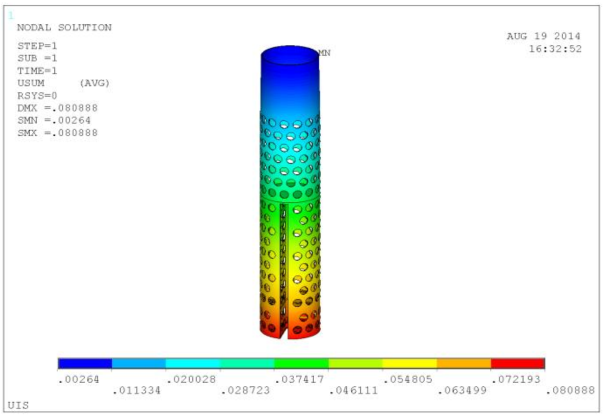
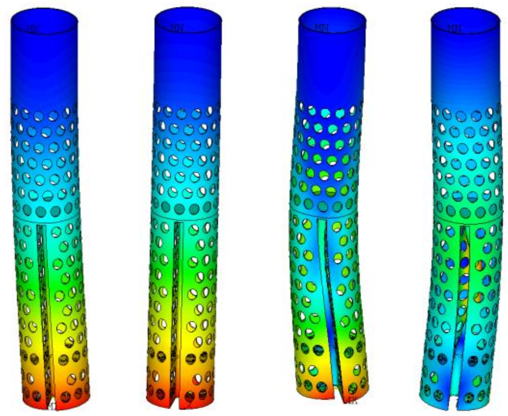


Fig. 9 Displacement distribution for the thermal load

2.3 Natural Frequency Analysis

The natural frequency analysis is performed to understand the characteristics of the vibration mode of the UIS. In the boundary conditions, all nodes of the top surface of the UIS are constrained in all directions.

As a result of the analysis, the dominant natural frequencies of the UIS are evaluated as shown in Fig. 10 and Table 1. In these figure and table, we can see that the main tube bending is the most dominant mode and the natural frequency is 13.29 Hz. The analysis result would be used in the basic data of the fluid added mass calculation and the development of the simple seismic analysis model.



(1st) (2nd) (3rd) (4th)

Fig. 10 Dominant mode shapes of the UIS

Table 1 Dominant mode characteristics of the UIS

Order	Natural Frequencies (Hz)	Mode Shape
1st	13.29	Shell cylinder structure 1st bending
2nd	13.78	Shell cylinder structure 2nd bending
3rd	51.55	Slot vertical plate 1st bending
4th	60.51	Slot vertical plate 2nd bending

3. Conclusion

The structural analysis model is developed to evaluate the structural integrity of the UIS. The primary stress analysis, the thermal stress analysis and the natural frequency analysis for the UIS are performed, and the maximum stresses and displacements are evaluated. From the analysis results, it is confirmed that the large local stresses don't occur near the holes and through the wall thicknesses of the structure. In addition, the maximum temperature of the UIS is calculated as 545 ^{o}C from the thermal analysis and the structure should be evaluated by the ASME design rules at a high temperature. In the future, the more detailed design will be performed by the high temperature evaluation procedure according to the ASME SEC. III, Div.5.

Acknowledgements

This study was supported by the MSIP through its National Nuclear Technology Program.

REFERENCES

[1] ANL, Advanced Burner Test Reactor Preconceptual Design Report, ANL-ABR-1, 2006.