Transactions of the Korean Nuclear Society Autumn Meeting
Busan, Korea, October 27-28, 2005

Stress Analysis of the Lower Core Plate Subjected to Radiation-Induced Heat Generation

S. H. Jeong, J. S. Moon, I. Y. Kim
Korea Power Engineering Company, Inc.
305-353, 150 Deokjin-Dong, Yuseong-Gu, Daejeon, Korea
E-mail : shjeong@kopec.co.kr

1. Introduction

The Lower Core Plate (LCP) is a perforated circular plate that supports and positions the fuel assemblies. The plate contains numerous holes to allow flow through the plate and provide flow to each fuel assembly. The plate is bolted at the periphery to a ring welded to the inside diameter of the core barrel. The center span of the plate is supported by the lower support columns which are attached at the lower end of the lower support plate. Temperature differences between components of the lower support assembly induce significant thermal stresses in the LCP. In addition, due to the LCP's proximity to the core, radiation-induced heat generation cause a significant temperature increase in this component. Stress analysis has been performed to evaluate structural integrity and the results are presented. Since the LCP has several small features, a technique referred to as submodeling is used to determine the stresses.

2. Stress Analysis

Finite element models were created using ANSYS [1] code. A coarse model and then a sub-model, with greater detail, were made. Using the models, a thermal analysis was performed to evaluate the temperature distribution throughout the LCP. Then, by applying the finite element temperature solution from thermal analysis, a structural analysis was done to determine the stresses. From the results of the coarse model and sub-model structural analyses, a fatigue evaluation was performed. Each of these processes is presented.

2.1 Analysis Model

A 1/8 symmetry coarse model was created. The coarse model includes the flow holes and a notch along the rim of the plate as shown in Figure 1. The thermal and structural element types are compatible, which allows the output of the thermal analysis to be read directly into the structural analysis.

2.2 Thermal Analysis

The temperature distributions throughout the LCP were calculated for steady state and transient conditions. Thermal loadings include convection from contact between the inlet reactor coolant and the exterior of the LCP and heat generation that act throughout the plate.

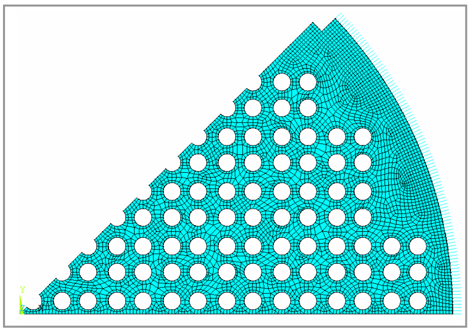

Figure 1. Coarse model of the LCP

The heat generation loadings were applied at the appropriate steady and thermal transient conditions at which high temperature gradients exist within the LCP. Thermal analyses were performed for four conditions, (1) full power, full flow, (2) steady state 118% overpower, (3) transient up envelope, and (4) transient down envelope.

Figure 2 shows distribution of full power heat generation rate, and Figure 3 shows resulting temperature distribution.

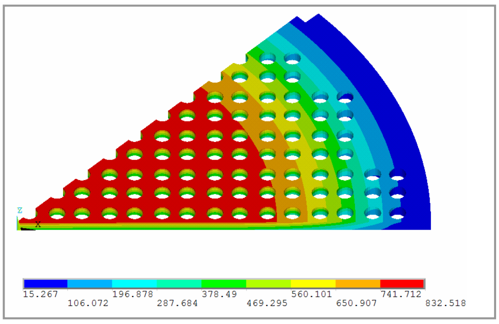

Figure 2. Full power heat generation rate

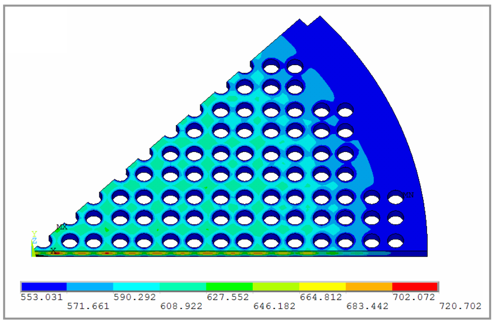

Figure 3. Temperature distribution at full power

2.3 Structural Analysis

Structural analysis was conducted using the same finite element model that was used in the thermal analysis. The loading includes the following:

(1) Temperature distribution derived from the thermal analyses
(2) Displacement boundary conditions at the support column locations restrain movement in the vertical direction,
(3) Boundary conditions along the cut edges (0° and 45°) will restrain movement in the direction normal to the cut surface.

The rim is connected with radial springs that simulate the stiffness of the core barrel at the location of the LCP. The total stiffness is the combination of the core barrel and the support ring, which LCP sits on, acting in series.

The stress intensities were determined for the same loading cases that were evaluated in the thermal analysis. The maximum stress intensities occurred in the middle of the perforated region as shown in Figure 4.

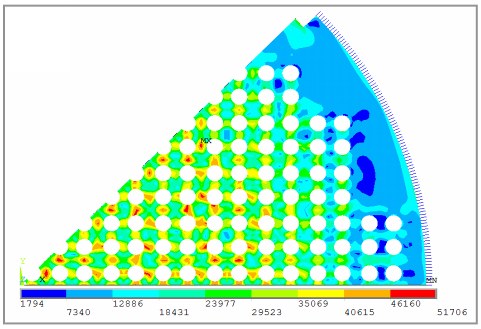

Figure 4. Stress intensities at full power, full flow

2.4 Assessment of Local Stresses

In the coarse model several small features were not included, which could affect the local stresses. These features include chamfers, column support bolt holes with chamfer, and alignment pin holes with counter bore. Based on these features, a typical section is selected and the analysis is done using an ANSYS feature known as submodeling.

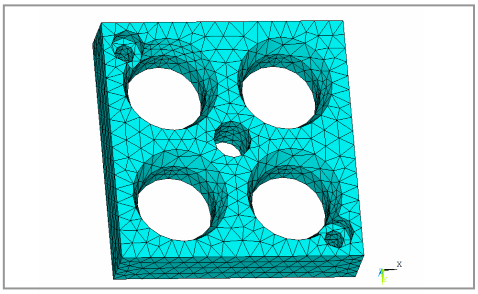

Figure 5. Sub-model of the LCP

With this technique, a sub-model is made which is spatially located within the coarse model. The boundaries of the sub-model are those surfaces that would be in contact with the coarse model. The displacement solution is transferred from the nodes of the coarse model to the corresponding surface nodes on the sub-model. The temperature loads of the coarse model are transferred to the sub-model at all nodes. The sub-model is shown in Figure 5. The maximum stresses occur in the alignment pin holes at the bottom of the counter bore as shown in Figure 6.

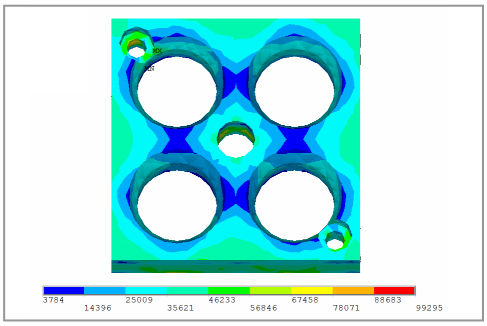

Figure 6. Stress intensity of the sub-model

2.5 Evaluation of Stresses

The stress analysis has determined plate stress intensities, including those stresses at local features. Now the LCP needs to be evaluated to the ASME Code, subsection NG [2]. In particular, the stress intensities were compared to the requirements for primary plus secondary stress intensity and fatigue as shown in Table 1.

Table 1. Summary of Results

Category	Range of Max. Stress (MPa)	Allowable Stress (MPa)	Margin of Safety
P_m+P_b+Q	300	335	0.117
Fatigue	-	-	U_{cum}=0.288

3. Conclusion

The stress analysis was done for the LCP subjected to radiation-induced heat generation. For fatigue evaluation, local stresses were calculated using ANSYS submodeling technique. Based on the analysis results, it can be concluded that the structural integrity of the LCP is maintained.

REFERENCES

[1]ANSYS User's Manual for Revision 7.1, ANSYS, Inc., 2003.
[2] ASME Boiler and Pressure Vessel Code, Section III, Division I, Subsection NG.