Transactions of the Korean Nuclear Society Spring Meeting
Gwangju, Korea, May 30-31, 2013

Preliminary Stress Analysis for Reactor Head of Prototype Sodium-cooled fast Reactor

In-su Han [a*], Gyeong-Hoi Koo [a] and Jae-Hun Cho [a]
[a] *Korea Atomic Energy Research Institute, 1045 Daedeok-daero, Yuseomg-gu, Daejeon, The republic of Korea*
[*] *Corresponding author: his0805@kaeri.re.kr*

1. Introduction

The sodium-cooled fast reactor (SFR) module consists of the vessel, containment vessel, head, rotating plug (RP), upper internal structure (UIS), intermediate heat exchanger (IHX), decay heat exchanger (DHX), primary pump, internal structure, internal components and reactor core. The reactor head forms the top closure for the reactor vessel. The reactor head provides various penetrations for an in-service inspection (ISI), maintenance, and fuel handling. In addition, it also supports the weight of several main components such as IHX, primary pump, UIS, CRDM, rotating plug, etc. A center of rotating plug is eccentric 200 mm away from the center position of the reactor head because IVTM (In-Vessel Fuel Transfer Machine) is able to handle all fuels in that position.

The purpose of this study is to evaluate the integrity of reactor head about the weight load of the main component acting on the reactor head.

2. Methods and Results

2.1 Analysis Condition

Material property of 316 stainless steel is applied to the reactor head FE model. For applying the load conditions, weights of main components are converted as pressures then they are applied to the area where they are installed. Only the uniform temperature of 150℃ is applied. Table 1 shows masses of the main components.

Table 1 Masses of the main components

component	quantity	Mass(kg)
Reactor head	1	82,960
Primary Pump	2	16,840
Upper Internal Structure	1	11,920
Intermediate Heat Exchanger	4	13,200
Decay Heat Exchanger	4	4,160
CRDM	10	240
In-Vessel Transfer Machine	1	5,200

2.2 Analysis Model

To perform the stress analysis, the 3-D finite element model was built by using the FEA software ANSYS[1]. Fig. 1 shows its 3D finite element model and Fig. 2 shows its loading conditions.

For the calculation, SOLID186 element was used, total 190,000 nodes and 40,000 elements were used. A numerical analysis is performed for the given condition.

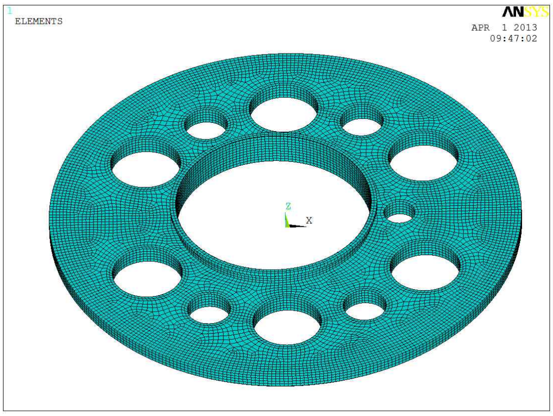

Fig. 1 Finite element model of reactor head

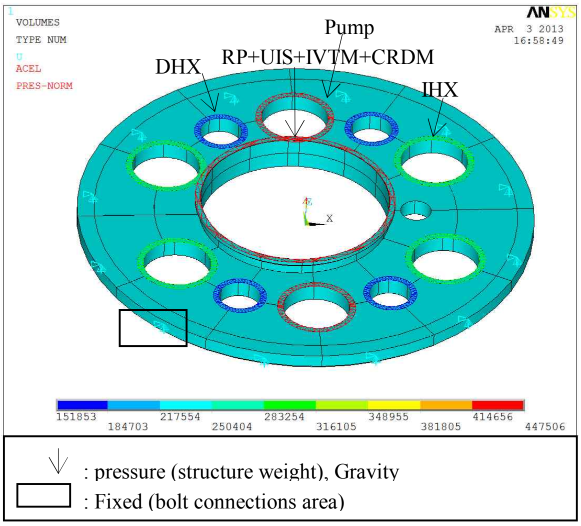

Fig. 2 Loading conditions of reactor head

3. Results and Discussions

3.1 displacement

Fig. 3 shows the displacement distributions of reactor head. The maximum displacement is 0.55mm in upper plate of the rotating plug support region.

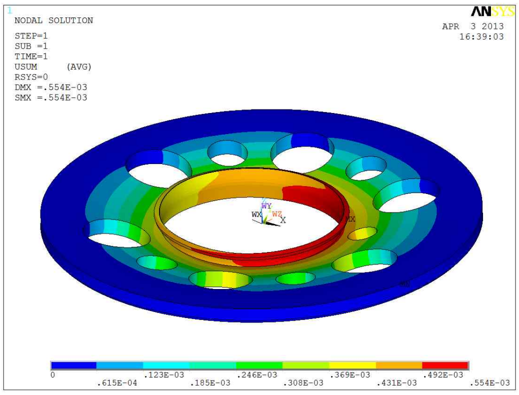

Fig. 3 displacement distribution of reactor head

3.2 stress result

Fig. 4 shows the stress intensity distributions of reactor head. The critical location is indicated in the maximum stress area in the rotating plug support region.

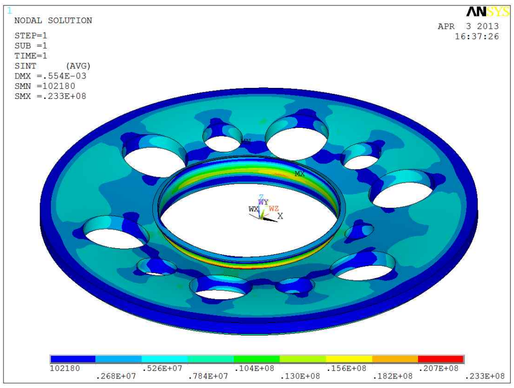

Fig. 4 Stress intensity distribution of reactor head

3.3 Stress Linearization

For evaluation in accordance with ASME Code, Stresses of cross section for stress evaluation are divided into the primary stress, secondary stress and peak stress [2, 3]. The results of stress linearization are compared with the allowable stress intensity.

In order to perform the stress linearization at the reactor head, four weak regions are selected where the maximum stress occurs, then the stresses are classified into the membrane stress and bending stress. Table 2 shows the calculated results. The analyzed results are compared with the allowable stress intensities of the material property. As a result, the maximum membrane+bending stress occurred on the end line of rotating plug support region, but all the stress intensity values are satisfied with the allowable limits [4].

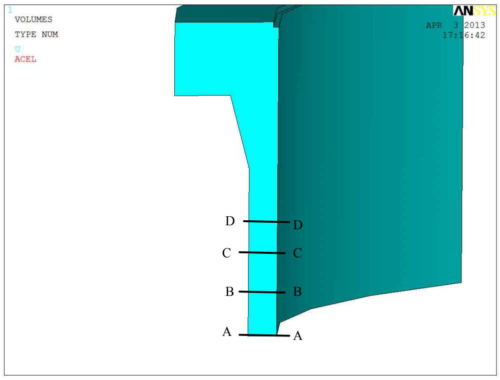

Fig. 5 Definition of the paths point for stress linearization

Table 2 Stress intensities of the respective paths

	Stress classification	Stress intensity [MPa]	Allowable limits (1.5Sm)[MPa]
A-A	$P_L + P_b$	18.9	172.5
B-B	$P_L + P_b$	15.6	172.5
C-C	$P_L + P_b$	12.9	172.5
D-D	$P_L + P_b$	8.9	172.5

4. Conclusions

In this paper, the stress analysis for the weight load of the main component acting on the reactor head was carried out. The maximum vertical displacement and stress intensity are obtained and it is confirmed that the stress intensities were satisfied with all the section about ASME code rule.

For the future work, the thermal and seismic analysis will be performed by combining the different design loads.

REFERENCES

[1] ANSYS, ANSYS Structural Analysis Guide, Version 14.5,2013.
[2] ASME Boiler and Pressure Vessel Code, Section Ⅲ, Division 1 – Subsection NB, ASME, 2007
[3] ASME Boiler and Pressure Vessel Code, Section Ⅱ, Part D, ASME, 2007
[4] Hyun-jin Park, Jae-Hun Cho, In-su Han, Ki-jong Jang and Tae-Wan Kim, Evaluation of Stress Intensity for SMART Steam Generator Tube Support Structure, The Korean Society of Mechanical Engineers Spring Meeting, May. 14, 2011