Stress Linearization of PZR Central Cover for SMART-P

Se-Hwan Lee, Gyu-Man Lee, Hong-Yune Park, Tae-Wan Kim, Jong-In Kim, Sung-Kyun Zee [a]
[a] Korea Atomic Energy Research Institute, Mechanical Engineering Division,
150 DukJin-Dong , YuSung-Gu, DaeJun, Korea, sehlee@kaeri.re.kr

1. Introduction

In general, stress evaluation criteria of nuclear power plant have been historically based on the ASME B&PV Code, Section III [1]. However, the PZR central cover of SMART-P has some difficulties in applying the design by formula or design by analysis method using a simple 2-dimensional finite element simulation. As the geometric parameters of central cover violate the conditions defined in the code for the application of perforated plate theory, Stress evaluation using a 3-dimensional finite element model is needed. Recently, the guidelines upon processing elastic finite element analysis results for the assessment of ASME Code stress limits are developing by the ASME subgroup on design analysis[2]. These guidelines discuss several recommendations about the stress evaluation method, especially, procedures for use in interpreting results of 3-dimensional elastic-finite element analysis and stress linearization procedures.

In this study, the stress linearization using 3-dimensional finite element analysis of the PZR central cover for SMART-P followed by the guidelines in Reference 2. In addition, the results of the 3-dimensional analysis are compared with those of the simplified 2-dimesinal stress analysis.

2. Analysis Model

The PZR central cover of SMART-P is an important structure because it preserves the pressure boundary as a rector vessel cover. The central cover is a circular plate with elliptical profile in the bottom. Also, it has many holes including the CEDM (control element derive mechanism) and small line nozzles. It could be classified as a non-typical perforated plate since the geometric conditions and numbers of holes do not meet the requirements depicted in the code. Then, the simplified equations derived in the code are not applicable to stress evaluation.

2.1 Material Property

The selected model of PZR central cover is made of SA240 type 321 (austenitic stainless steel). The adopted temperature dependent thermal and mechanical properties of all materials, which are referenced from ASME B&PV Code [3], were considered in numerical analysis. Table 1 represents the temperature dependent mechanical properties that are applied in finite element analysis.

Table 1. Mechanical properties of SA240 type 321

Temperature (℃)	Mechanical material properties		
	Elastic Modulus (GPa)	Thermal Expansion Coeffi.$\times 10^{-6}$ (mm/mm °C)	Poisson's Ratio
21.1	195	15.2	0.267
93.3	190	16.4	0.283
200.4	183	16.8	0.290
315.6	174	17.1	0.310

2.2 Finite Element Analysis

The finite element analyses are done to determine the temperature and stress field under the design and operation condition. Using the results of the thermal analysis, the thermal stress analysis during normal operation condition is performed.

The 3-dimensional finite element mesh generation and applied boundary conditions for the design and operation condition (thermal) are shown in Figure 1. The adopted FE model is a half model under consideration of symmetric behavior. Since the cooler to maintain the temperature of CEDM to certain level is designed, the temperature boundary conditions are applied as shown in the Figure 1 including the CEDM cooler temperature.

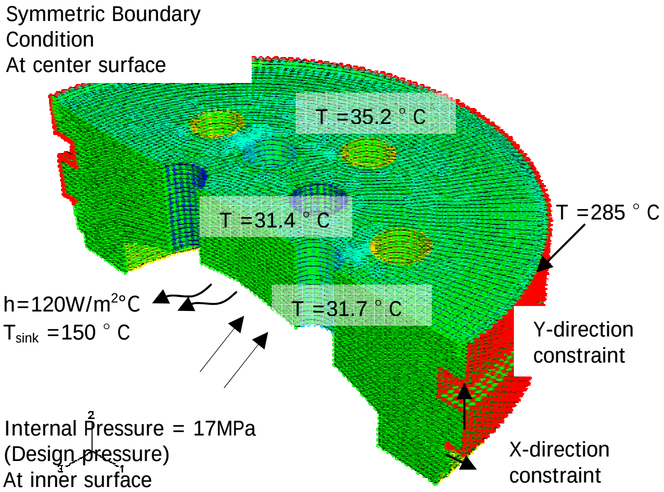

Figure 1. Finite element mesh and boundary conditions

2.3 Stress Linearization

The results of the elastic stress analysis for each condition (design and normal operation) are used for the stress linearization. First, the stress classification line (SCL) and stress classification plane (SCP) which is expected to produce higher membrane and bending stresses are defined in accordance to the guidelines in Reference 2. Figure 2 shows defined stress classification lines and planes.

SCL 1 to 5 in the Figure 2 are selected at the symmetry plane and two SCPs are chosen at the ligament between the CEDM holes to verify the stress limits. Calculation of membrane and membrane-plus-bending stress intensities from detailed total stresses are performed by ABAQUS [4]. The typical results of the stress linearization at each SCL are shown in Table 2. The stress results should meet the ASME Code stress limits given as following equation (1)-(3) for design and normal operation condition.

Design Condition:

$$P_L < 1.5\ S_m \tag{1}$$

$$P_L + P_b < 1.5\ S_m \tag{2}$$

Normal Operation Condition:

$$P_L + P_b + Q + F < 3\ S_m \tag{3}$$

(total stress)

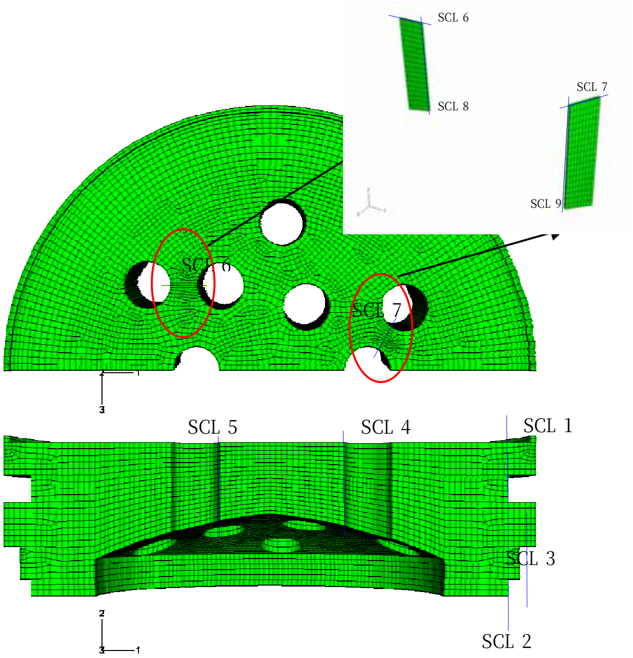

Figure 2. Stress classification lines and planes.

Table 2. Results of the stress linearization for SCLs (normal operation condition)

SCL	Total Stress (MPa)	3 Sm (MPa)
SCL 1	82.68	378.51 (at 315.56°C)
SCL 2	33.23	
SCL 3	111.48	
SCL 4	131.18	
SCL 5	118.69	
SCL 6	89.00	
SCL 7	80.34	
SCL 8	90.41	
SCL 9	73.35	

From the results of stress linearization at each SCL, SCL 4 has the largest stress level but it has an enough margin. The calculated stress results of a 2-dimensional FE model expect the same trend but slightly higher value than the results of 3-dimensional model is reported.

3. Conclusion

3-dimensional finite element analyses are carried out to calculate stress value and integrity of PZR central cover of SMART-P. The purpose of the 3-dimensional finite element analyses was to assess and verify the stress distribution of central cover under design and normal operation condition according to the new guidelines of ASME code.

These calculations results showed that the stresses due to design pressure of 17 MPa and the temperature difference between inside and outside the cover are within the allowable limits imposed by the ASME Code stress limit.

Acknowledgment

This project has been carried out under the nuclear R&D program by MOST.

REFERENCES

[1] ASME Boiler and Pressure Vessel Codes, Sec. III, Nuclear Reactor Vessels, American Society of Mechanical Engineers, NY, 1998

[2] Subgroup on Design Analysis, Proposed Non-Mandatory Appendix for Section III & VIII : Guidelines on Processing Elastic Finite Element Analysis Results for Assessment of ASME Code Stress Limits, Rev. 15, 2003.

[3] ASME Boiler and Pressure Vessel Codes, Sec. II Part-D, Nuclear Reactor Vessels, American Society of Mechanical Engineers, NY, 1998.

[4] ABAQUS User's Manual, Ver. 6.3, 2003