Structural Analysis of a Reservoir Tank in the STELLA-1 Test Facility

S. H. Kim [a*], Y. B. Lee [a], H. Y. Lee [a], H. Y. Park [b]
[a]*Korea Atomic Energy Research Institute, Daejeon 305-600, The Republic of Korea*
[b]*AD-Solution Co., Ltd., Daejeon 305-301, The Republic of Korea*

[*]Corresponding author: *shkim5@kaeri.re.kr*

1. Introduction

The sodium integral effect test loop for safety simulation and assessment(STELLA-1) is a large scale separate effect test facility for the thermal hydraulic performance verification for heat exchangers of passive decay heat removal circuit and the mechanical sodium pump of the primary heat transport system(PHTS) which is the significant component for the safety verification in the sodium cooled fast reactor(SFR). The test facility is composed of the main experimental system, the auxiliary system and the data acquisition system including the instrumentation and control system. A reservoir tank performs the function absorbing the thermal expansion during the closed circuit operation of the high temperature loop and the experimental loop of the PHTS pump in STELLA-1 and the role of the buffer tank for sodium. The installation position of the reservoir tank is located in the upper part of the STELLA-1 facility as shown in Fig. 1, which performs the function to discharge Ar gas inside the loop during sodium injection. To prevent the solidification of sodium, the electric heater which can heat up the room temperature to the preheating temperature of 150 ^{o}C~200 ^{o}C in 25 ^{o}C/hr and maintain the inside sodium temperature considering the surrounding heat loss is attached in the outer wall of the tank and controls the temperature. The design parameter of the reservoir tank is presented in Table 1[1].

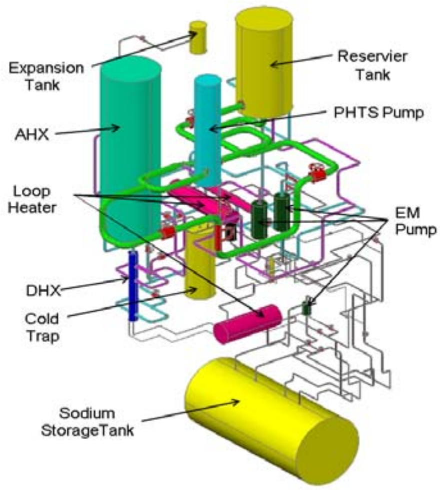

Fig. 1 Installation position of the reservoir tank in STELLA-1

Table 1 Design parameter of the reservoir tank

Item	Content
Material	STS 316/ STS 316L
Design Temperature	600 ^{o}C
Design Pressure	1.0 MPa(A)
Operating Condition	RT~510 ^{o}C/ ~ 1.0 MPa(A)

2. Structural Analysis of a Reservoir Tank

As shown in Fig. 2, the piping and the pressure control valves connected to the gas supply system and the exhaust system are installed in the upper part of the reservoir tank. The nozzle pipe and the support leg of the tank are used for the material of SS316 and the shell and the plate are made for the material of SS316L.

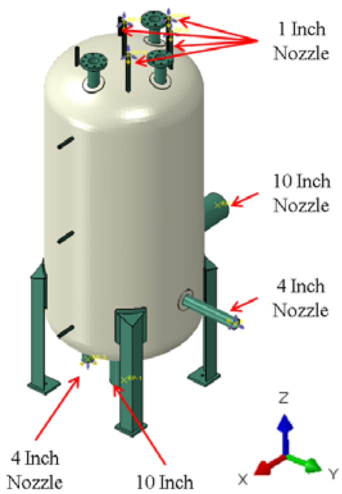

Fig. 2 3D cad configuration of reservoir tank

2.1 FE Model for the Analysis

The following finite element model based on the 3D cad configuration is modeled by using the ABAQUS program as shown in Fig. 3. The loads for the primary stress analysis consider the nozzle load, the dead weight including the sodium inside the tank and the design pressure. The boundary conditions of the support legs which are the significant factor for the occurrence of the extreme stress concentration does not consider the rigid coupling and apply three types of constraints such as the x and y directional sliding condition and xy plane sliding condition. The thermal load for the secondary stress analysis is applied for the uniform temperature of 600 ^{o}C which does not consider the transient condition because the outside of the model is insulated. Also, the nozzle load for the secondary stress analysis is applied .

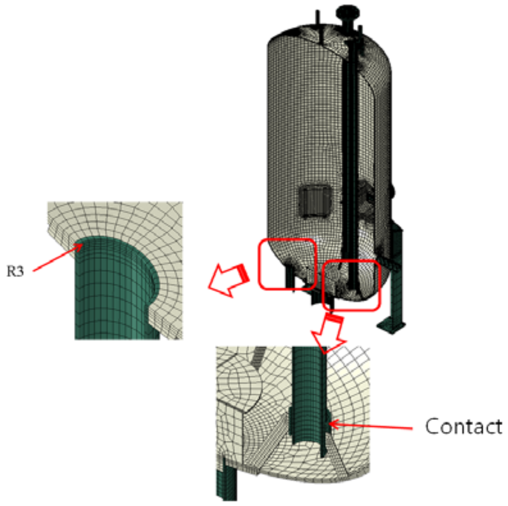

Fig.3 Finite element model for the 3D configuration

2.2 Results of the Structural Analysis

The analysis results for primary and secondary stresses are shown in Fig. 4 and Fig. 5, respectively. In the case of primary stresses, the maximum stress occurs in the shell and the reinforcement pad of the N2 nozzle attached to the left side of the tank as shown in Fig. 4. In the secondary stresses, it can be seen that the maximum stress occurs in the neck pipe of the N1 nozzle attached in the bottom side as shown in Fig. 5. The structural integrity of the reservoir tank is evaluated by ASME Sec. VIII Div. 2 and material properties apply the ASME Sec. II, Part D[2,3]. The contact part connecting the support leg and the reinforcement pad creates the local stress concentration, but such geometrical discontinuity can be mitigated by the fabricating process. Table 2 shows evaluation results of structural integrity of the reservoir tank in the STELLA-1.

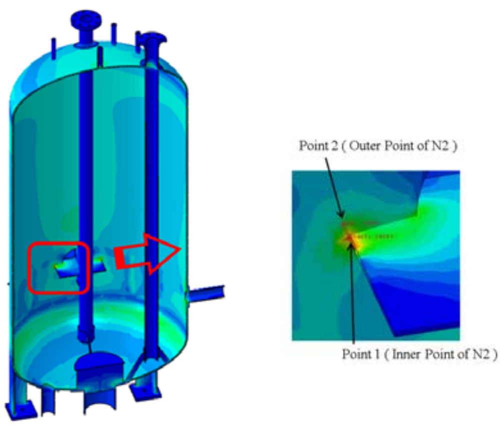

Fig. 4 Stress analysis results for primary loads

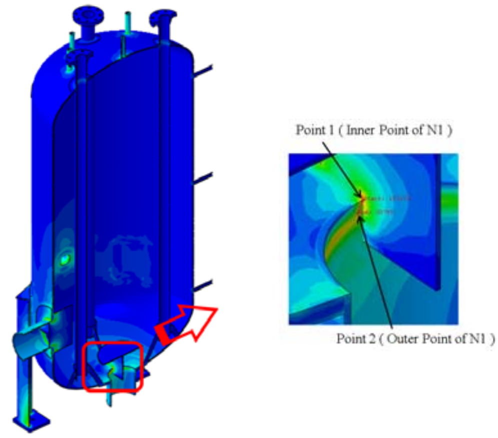

Fig. 5 Stress analysis results for secondary loads

As shown in this table, it can be seen that the stress evaluation results satisfy the stress allowable limits

Table 2 Evaluation results of structural integrity

Evaluation Items	Calculated Value(MPa)	Allowable Limit(MPa)
❑ Primary Stress Limits		
Membrane	58.8	121.2
Membrane + Bending	114.1	121.2
❑ Primary + Secondary Stress Limits		
Membrane + Bending + Secondary Stress	115.3	242.4

3. Conclusion

From the evaluation results according to the allowable design limits of ASME Sec. VIII Div. 2, the structural integrity for the reservoir tank is confirmed in the given design condition.

Acknowledgements

This study was supported by the Korean Ministry of Education, Science and Technology through its National Nuclear Technology Program.

REFERENCES

[1] T. H. Lee, et al., Design Report of STELLA-1, KAERI/TR-4295/2011, Korea Atomic Energy Research Institute, 2011.
[2] ASME Boiler and Pressure Vessel Code Section VIII, Div. 2, ASME, 2004.
[3] ASME Boiler and Pressure Vessel Code Section II, Part D, ASME, 2004.