THERMAL HYDRAULIC ANALYSIS ON THE TRANSIENT PHENOMENA IN THE PRESSURE BALANCING LINE OF THE HYBRID SAFETY INJECTION TANK

Sungje Hong, Kunwoo Yi , Jin Huh, Dohyun Kim, Joohee Lee,
Inyoung Im, Sungchan Park and Changkyu Chung
NSSS Division, KEPCO Engineering and Construction Company INC.,
989-111 Daeduckdaero, Yuseong-gu, Daejeon, Korea
sjhong@kepco-enc.com; kwyi@kepco-enc.com; jhuh@kepco-enc.com; dohkim20@kepco-enc.com; elevenfinger@kepco-enc.com; iyim@kepco-enc.com; scpark@kepco-enc.com; ckchung@kepco-enc.com

ABSTRACT

The Hybrid Safety Injection Tank (H-SIT) is one of the main components of the passive core cooling system that can be used for both high and medium pressure safety injections. The top of the H-SIT is connected to the cold leg through the Pressure Balancing Line (PBL) and normally separated from the Reactor Coolant System (RCS) by two isolation valves on the PBL. When the valves open for high pressure injection, very complex thermal hydraulic phenomena, such as rapid steam condensation and turbulent mixing, are expected inside the PBL and the H-SIT. In this study, CFD analysis is performed in order to examine the thermal hydraulic phenomena at the moment of opening the PBL isolation valve in the PBL during the accident requiring high pressure safety injection. As results of the calculation, three propagation parts in series appeared and pressure jumps and supersonic flow areas were observed. The highest pressure peak increased by 5 times of RCS pressure at the center of the front propagation part. These variation of the parameters happened within 0.1 second. The following computational models are defined for the analysis; three dimensional, implicit unsteady, coupled flow, Eulerian multiphase, SST k-Omega turbulence, and gravity.

KEYWORDS
CFD, Hybrid, SIT

1. INTRODUCTION

Safety has been a critical issue in the nuclear industry. Since after Fukushima disaster in 2011, the issue has emerged with growing global interest. Passive safety system is considered as one of the inherent safety concepts due to its passive operating methodology and provides several benefits from various aspects of operation or installation such as cost effective installation, convenience in maintenance, and operability. In recent years, the use of the passive safety systems such as accumulator and gravity driven injection system is motivated with the potential for increased reliability of the safety systems[1].

The Hybrid Safety Injection Tank (H-SIT) is one of the main components of the passive core cooling system, which is filled with water and nitrogen to be used for both high and medium pressure safety injections. The top of the H-SIT is connected to the cold leg through the Pressure Balancing Line (PBL) and normally separated from the Reactor Coolant System (RCS) by isolation valves on the PBL. The H-SIT is pressurized with nitrogen gas at a medium pressure and provides safety injection to reflood the core in a large break loss of coolant accident (LBLOCA). When the isolation valves in the PBL are opened by the actuation signal for the high pressure safety injection in high pressure accidents such as the total loss of feedwater, the high pressure RCS water provides an equalizing pressure through the PBL to the H-SIT and boric acid water in the H-SIT is injected into the reactor vessel through the Direct Vessel Injection (DVI) nozzle by gravity force and pressure difference.

When the isolation valves open for high pressure injection, very complex thermal hydraulic phenomena, such as rapid steam condensation and turbulent mixing, are expected inside the PBL and the H-SIT. The phenomena of condensation and turbulent mixing are critical to the function of the H-SIT from the perspective of initiation of injection and pressure buildup in the H-SIT. Therefore, this study is performed to investigate thermal hydraulic phenomena in the PBL during the accident requiring high pressure safety injection, using a 3D Computational Fluid Dynamics (CFD) code.

2. NUMERICAL SIMULATION

2.1. Configuration of Geometric Model

For computational analysis, the piping and the isolation parts from the PBL to the H-SIT discharge nozzle are modeled. The isolation valves in series on the PBL are simplified as an interface boundary between water and gas so that the quickly opening of the valves can be simulated. Schematic flow diagram of the H-SIT is shown in Figure 1.

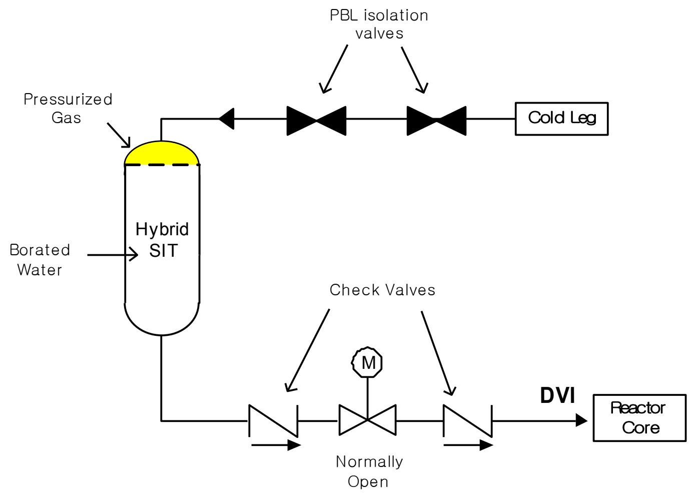


Figure 1. Schematic Flow Diagram of H-SIT

2.2. Modeling approach

The following computational models are used for the analysis; Implicit Unsteady, Coupled Energy, Exact wall Distance, SST (Menter) K-Omega turbulence model, Coupled Flow, Multiphase Equation of State, Two-Phase Thermodynamic Equilibrium, Eulerian Multiphase Mixture, Multiphase Interaction, Eulerian Multiphase, Three dimensional equations, and Gravity.

The three-dimensional Navier–Stokes and energy equations are employed. Two-equation turbulence models are applied in this study considering efficiency and accuracy of the calculation, although there are various turbulence models such as one-equation models, large eddy simulation (LES), and Reynolds stress model (RSM). Because SST K-Omega turbulence model can predict shock waves, mixing layers and jet flow problems properly, the turbulence model is used [5, 6].

Over four million polygonal meshes which are generated by the STAR-CCM+ [5] are used with a quality threshold of 0.4. When compared to a tetrahedral or hexahedral cell, a polyhedral cell has more faces, and therefore it has more optimal flow directions (normal to a face) than a tetrahedral or hexahedral cell. The polyhedral cells have more neighbors which allows better gradient approximations, especially near boundaries and corners. Base size is determined as 2 inch considering efficiency of calculation for geometric model which had a ratio of about 0.023 in inner diameter between PBL and H-SIT. However, the meshes in the PBL have a base size 10% relative to the tank and 1.25% in certain areas. The expanded area at the end of PBL connected to the tank is meshed with a base size 20% relative to the tank. Geometric model of the H-SIT, computational grid and cross section of the grid system are shown in Figure 2, Figure 3, and Figure 4.

Figure 2. Geometric Model of the H-SIT

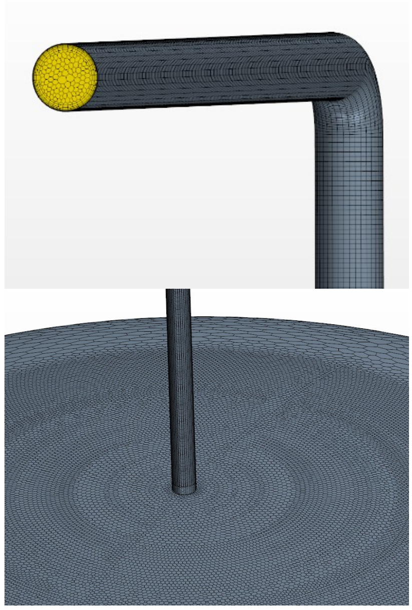
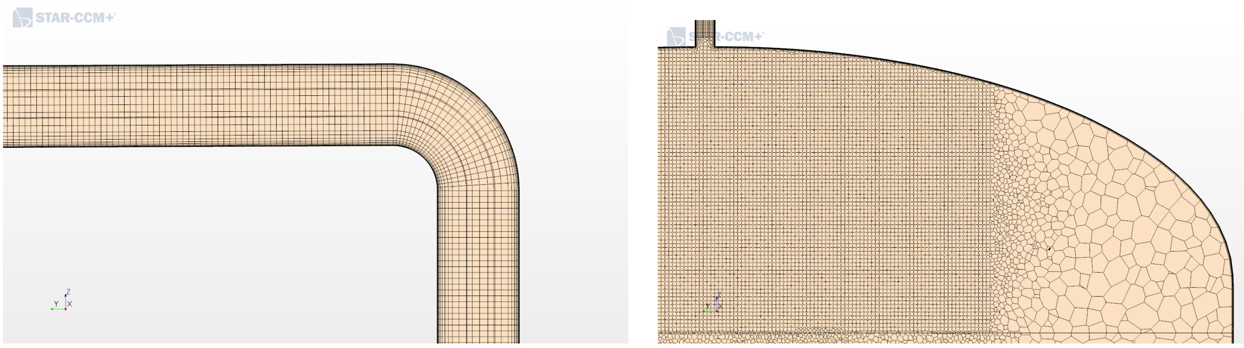


Figure 3. Computational Grid of PBL and H-SIT



(a) Pressure Balancing Line (PBL)

(b) Hybrid Safety Injection Tank (H-SIT)

Figure 4. Grid System in (a) PBL and (b) H-SIT

Minimum face quality of 0.05 is given for the surface. Eight prism layers are used for the walls with a stretching factor of 1.5 in about 0.2 inch of prism layer thickness. Maximum safe skewness angle is set to 75 degree, while minimum unsafe skewness angle is 88 degree.

2.3. Boundary Conditions and Simulation Parameters

The RCS and the H-SIT are assumed to be at the normal operation conition, initially. The modeling area is separated into three regions; water region in the PBL which is from branch line of the RCS to the upstream of the isolation disk, pressurized gas region of the PBL and H-SIT which is from the downstream of the isolation disk to the water surface of the H-SIT, and water region of the H-SIT which is from water surface to discharge outlet of the H-SIT. The working fluids are assumed as water for liquid medium and steam for pressurized gas medium. The initial and boundary conditions of the RCS, PBL, and H-SIT are listed in Table I.

The inlet flow from the reactor has a fully developed profile with a temperature of 555 °F in the PBL. The outside of the PBL is assumed to be adiabatic. The outlet boundary of the H-SIT is regarded as a wall in order to investigate transient thermal hydraulic phenomena during the buildup time for pressure stabilization. Properties of water and steam are calculated as a function of pressure and temperature in the International Association for the Properties of Water and Steam-Industrial Formulation 1997 (IAPWS-IF97).

Table I. Initial and Boundary Conditions of the H-SIT analysis

Parameters	Value	Unit
Inlet Pressure	2,250	psia
Inlet Temperature	555	°F
Pressure in SIT	610	psig
Temperature in SIT	120	°F
Outlet Pressure	Wall condition	

2.4. Results and Discussion

CFD simulations are performed using a commercial code, STAR-CCM+ [5]. The case presented here, and the other figures in this sub-section, is the pipe of diameter 2.5 inch and the inlet pressure is at 2,250 psia. The CFD results show the distribution of the thermal hydraulic parameters variation at 0.02 second. Total 80 measurement points, 40 points per each side of the isolation disk part with the water and the gas regions, are allocated with equal intervalalong the line of top, center, and bottom of the PBL.

As the PBL isolation valve is quickly opened, the pressure difference in the gas region, between PBL isolation valve and end of the PBL, fluctuates but increases, and it becomes stabilized with time because the lower pressure region is filled with compressible fluid, steam, as shown in Figure 5.

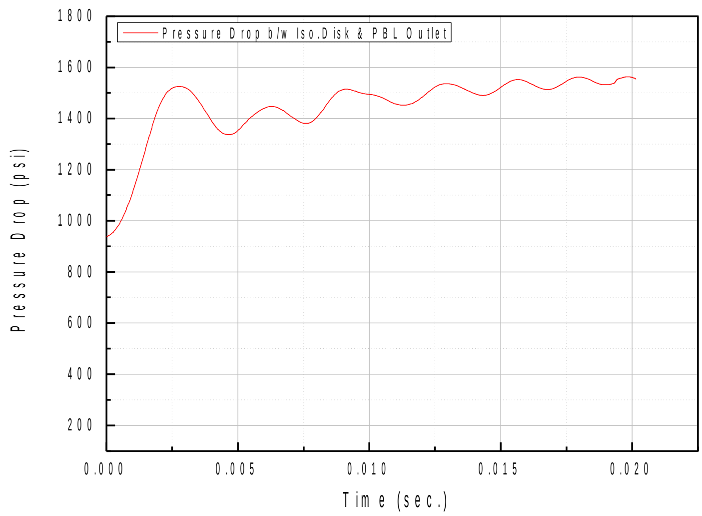


Figure 5. Pressure Drop in the Gas Region of the PBL

Figure 6 and Figure 7 show the pressure and temperature profile throughout the PBL. It can be seen that the highest pressure peak, maximum about 11,740 psi which is over the RCS design pressure, is observed at the center and near the pipe wall of the front part of propagation and very low pressure area (about 610 psi) followed after the front part of the propagation at the vicinity of the pipe wall. The highest temperature, maximum about 2,490 °F, is observed at the center of the front part of the propagation as shown in Figure 8 and Figure 9.

Figure 8 and Figure 12 show values of total pressure and Mach number at the measurement points of top, center, and bottom of the PBL. There are series of two pressure and velocity propagation parts which are separated with lower pressure and velocity areas near the pipe wall. While the pressure on the front part of propagation is calculated higher than that on the latter part of propagations, there are no significant differences between front and latter parts of velocity. The supersonic flow is developed with maximum Mach number of about 1.3 and two of shock wave propagations are observed. When a supersonic flow passes through a pipe, it forms a combination of shocks and expansion waves called shock train [7]. The shock train propagates along the PBL wall and causes fluctuations of the flow properties especially in the static pressure and temperature which are observed in Figure 8 and Figure 9.

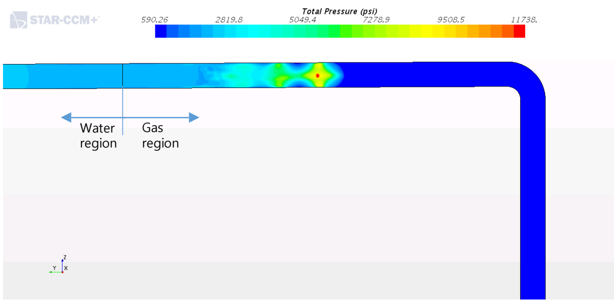


Figure 6. Pressure Profile in the PBL

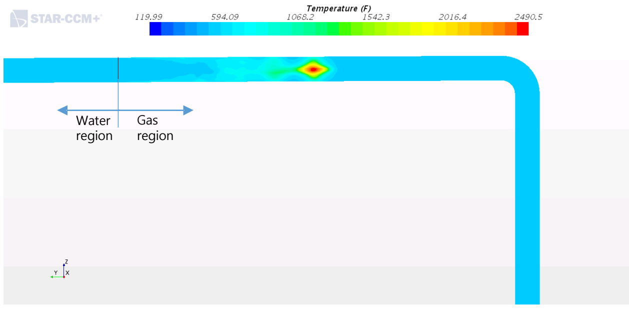


Figure 7. Temperature Profile in the PBL

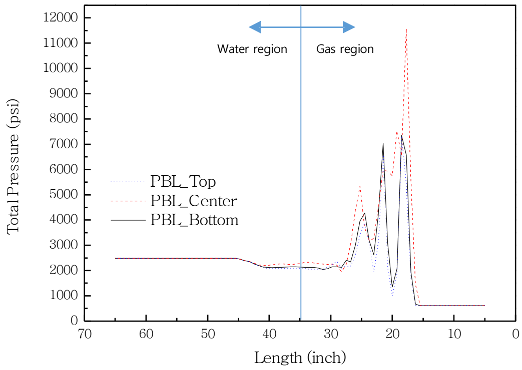


Figure 8. Total Pressure vs Length of the PBL

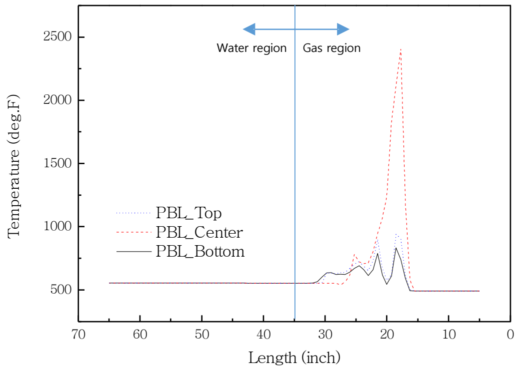


Figure 9. Temperature vs Length of the PBL

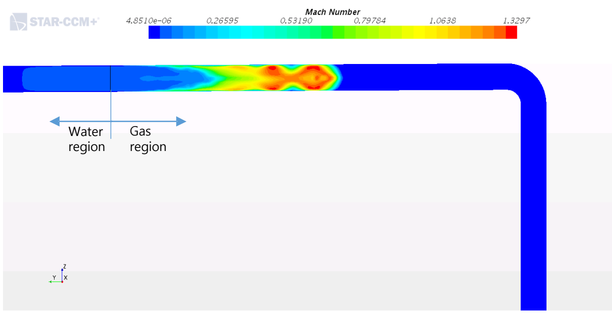


Figure 10. Mach Number

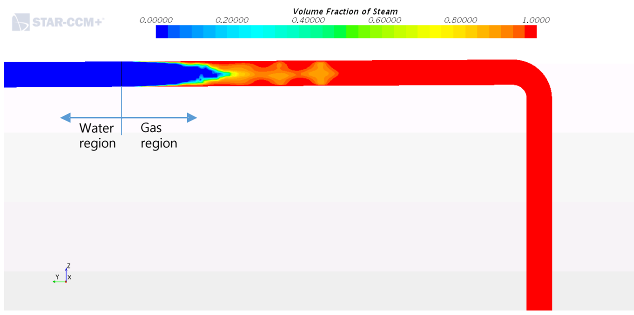


Figure 11. Volume Fraction of Steam

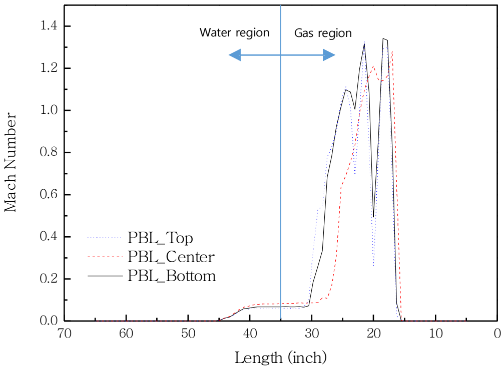


Figure 12. Mach Number vs Length of the PBL

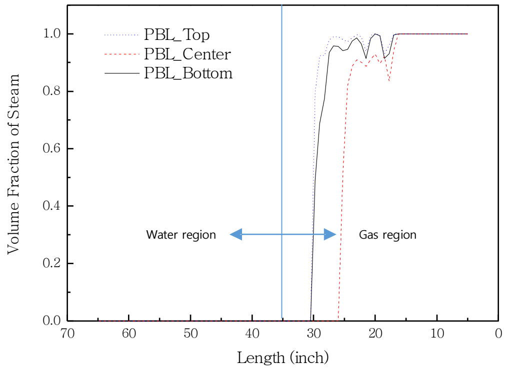


Figure 13. VOF of Steam vs Length of the PBL

Figure 11 shows profile of volume fraction (VOF) of steam and Figure 13 indicates values of VOF of steam along the PBL. There are also two condensation areas in the gas region which were induced by local pressure and temperature condition.

Three small or large peaks of total pressure, temperature, Mach number, and VOF of steam are observed at same locations of the PBL. At the locations where the highest pressure are measured in three propagation parts, the highest temperatures and the Mach numbers are generated, respectively, and condensation occurred at the three same points. The peaks of the thermal hydraulic parameters are observed at a same interval of distance.

3. CONCLUSIONS

This study was performed to investigate thermal hydraulic phenomena in the PBL of the H-SIT when the PBL isolation valve is quickly opened. As the water in the RCS (which is higher pressure region) flows into the H-SIT (which is lower pressure region), several pressure peaks were observed and supersonic flow areas appeared at the locations where the pressure peaks occurred. The highest pressure was observed at the center and near the pipe wall of the first pressure wave and temperature was increased up to maximum 2,490 °F at the center of the first pressure wave. The pressure peak increased by 5 times of the RCS pressure and the maximum Mach number was calculated as about 1.3 where the highest pressure appeared. The peaks of the thermal hydraulic parameters were generated at same locations with same intervals. These variation of the parameters happened within 0.1 second.

Since the quickly open operation of the isolation valve in the PBL, which has pressure difference between upstream and downstream of the isolation valve by over 3 times of downstream pressure, can make severe transient environment, various parameters, such as stroke time or operator type of valve, should be considered for choosing isolation valve.

ACKNOWLEDGEMENTS

The work described herein was performed by the financial support from the Ministry of Trade, Industry & Energy (MOTIE) of the Republic of Korea and the Korea Hydro & Nuclear Power Company (KHNP) in accordance with the Contract for the Development of the Design Requirements for the Hybrid SIT and the Passivie Emergency Core Cooling Systems between KHNP and KEPCO-E&C.

4. REFERENCES

1. IAEA-TECDOC-1624, Passive Safety Systems and Natural Circulation in Water Cooled Nuclear Power Plants
2. Bartosiewicz, Y., Aidoun, Z., Mercadier, Y., 2006. Numerical assessment of ejector operation for refrigeration applications based on CFD. Appl. Thermal Eng. 26, 604-612
3. Giancarlo Giacchetta, Mariella Leporini, Barbara Marchetti, Alessandro Terenzi, “Numerical study of choked two-phase flow of hydrocarbons fluids through orifices,” Journal of Loss Prevention in the Process Industries 27 (2014) 13-20
4. A. Attou, L. Bolle, J.M. Seynhaeve, “Experimental study of the critical flashing flow through a relief line: evidence of the double-choked flow phenomenon,” International Journal of Multiphase Flow 26 (2000) 921-947
5. “STAR-CCM+, Version 8.02 USER GUIDE,” www.cd-adapaco.com
6. G. Arnulfo, C. Bertani and M. De Salve, “Prediction of two-phase choked-flow through safety valves,” 31st UIT (Italian Union of Thermo-fluid-dynamics) Heat Transfer Conference 2013, Journal of Physics: Conference Series 501 (2014) 012016
7. Kazuyasu Matsuo, Yoshiaki Miyazato, Heuy-Dong Kim, “Shock train and pseudo-shock phenomena in internal gas flows,” Progress in Aerospace Sciences, 35, 1999, 33-100