Transactions of the Korean Nuclear Society Autumn Meeting
Changwon, Korea, October 20-21, 2022

CFD Simulation of Thermal Hydraulic Phenomena After Break

Sung Gil Shin [1*], Hansik Jung [2], Jai Oan Cho [1], Jeong Ik Lee [1*], Minki Cho [2], Jong-in Kim [2]
[1] *Dept. Nuclear & Quantum Eng., KAIST*
291, Daehak-ro, Yuseong-gu, Daejeon, 34141, Republic of Korea
[2] *Doosan Enerbility Co Ltd., Nuclear Power Plant BG, Steam Generator Design Team*
22, Doosanvolvo-ro, Seongsan-gu, Changwon, 51711, Republic of Korea
[*] *Corresponding author: jeongiklee@kaist.ac.kr*

1. Introduction

The break accident is regarded as one of the important limiting events in a pressurized water reactor (PWR) safety analysis. When the break accident occurs, it is necessary to maintain the integrity of the internal structures by the shock wave. In order to properly evaluate the maximum force applied to internal structures, it is necessary to analyze the flow field in three dimensions at early stage of the break [1]. In this study, hypothesis break accident is analyzed with a computational fluid dynamics (CFD) code, ANSYS CFX 19.2.

2. Methods and Results

2.1 Modeling Method

Fig. 1. shows the calculation domain and mesh in CFX. A tank simply simulates a pressure vessel, and a long nozzle simulates ambient volume which the break flow is discharged. For reducing computational domain, only half of the entire domain was modeled using the symmetry condition. For high mesh quality, sweep and hex dominant method was utilized for meshing. Detailed mesh information is summarized in Table I.

(a)
(b)
(c)

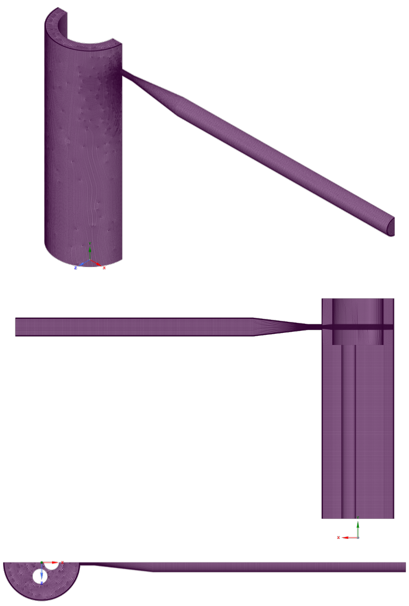


Fig. 1. Geometry and Mesh Information (a) orthographic view (b) back view (c) top view

Fig. 2. shows boundary conditions. The inside of the tank is set at 6.89 MPa and 232.22 ℃ of subcooled liquid state, and the nozzle is set to steam at 1 bar. A static pressure condition of 1 bar is assumed to the end of nozzle, wall of tank and nozzle are assumed adiabatic condition. The tank wall is set to have no-slip condition, and the nozzle wall is in free slip condition. This is because it is predicted to have a very high fluid velocity at the nozzle with very thin boundary layer. In order to model the phase change around the water jet at the break, it is important to select an appropriate multiphase flow model. Eulerian model was used to model the two-phase flow. Since the phase of discharged water jet changes to the dispersed phase quickly, water is assumed to be continuous fluid and vapor to be dispersed fluid. Additionally, selected models are summarized in Table I.

Table I: Modeling Information

Mesh information	
Method	Sweep/ Hex Dominant/ Layers
Mesh size	0.01 m
Number of nodes	757,346
Number of elements	1,332,632
Modeling information	
Fluid models	2C, 6M, 2E, 1V
Turbulence	SST
Multiphase	Particle
Dispersed phase mean diameter	0.1 mm
Interphase momentum	Drag (Schiller-Naumann)
Interphase mass	Thermal phase change
Interphase heat	Hughmark
Material Properties	IAPWS library
Analysis type	
Analysis type	Transient
Total time	0.02 sec
Timestep option	Adaptive (max Courant no.)
Max Courant number	1.0
Residual target	1E-4
Solver information	
Transient scheme	First order backward Euler
Advection scheme	Upwind
Turbulence numerics	First order

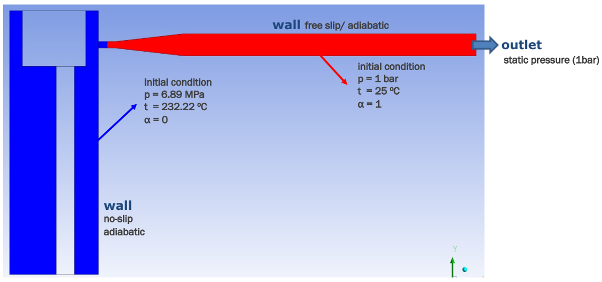
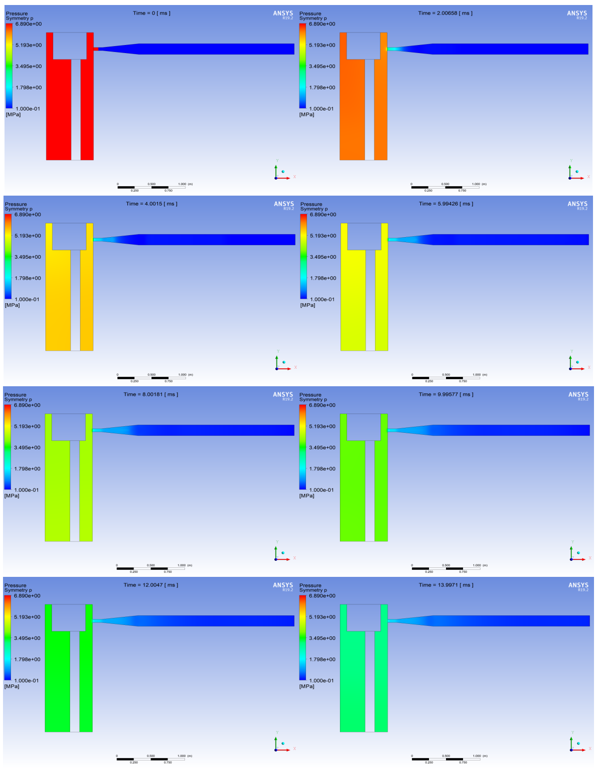


Fig. 2. Boundary Conditions.

2.2 Results

Fig. 3. shows the pressure change of the calculation domain with 2 msec interval after the break, and Fig. 4. shows the change of void fraction with the same interval. It can be seen that the tank pressure decreases with time as the water jet is discharged from the tank. Fig. 6. shows the pressure change at each position shown in Fig. 5., and Fig. 7. shows the mass flow rate released from the break. From the pressure contours and graph, it can be seen that the tank pressure rapidly decreases as the pressure wave propagates for 20 msec. Meanwhile, Fig. 8. and Fig. 9. show RELAP5 analysis results conducted by Doosan Enerbiltiy for the same geometry and conditions [2]. When Fig. 6. and Fig. 7. are compared, it can be seen that the results obtained from CFX and RELAP5 have good similarity.



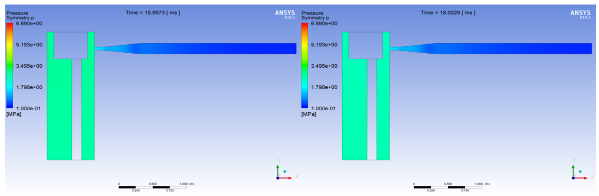


Fig. 3. Pressure Changes (every 0.2 milliseconds).

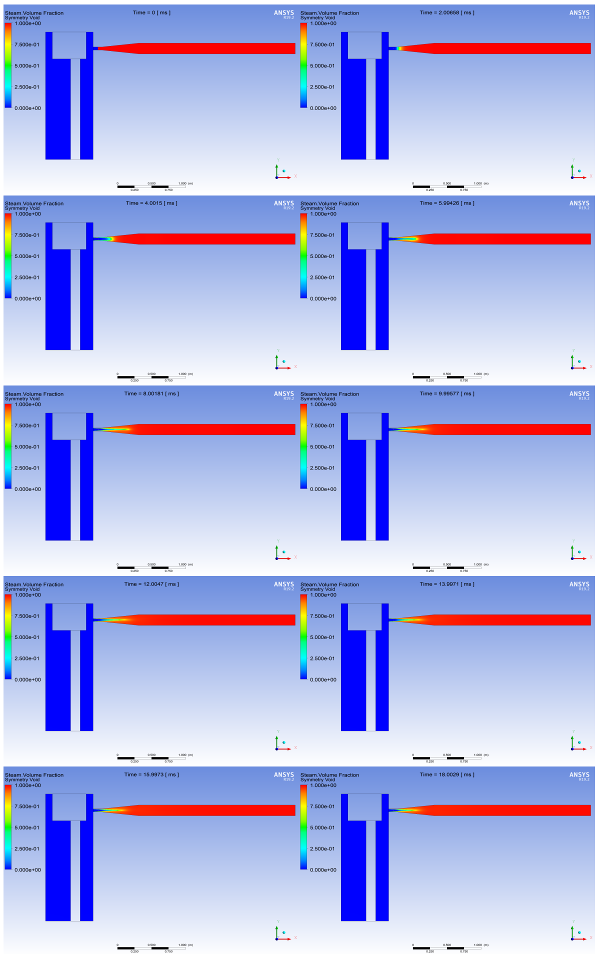


Fig. 4. Void Fraction Changes (every 0.2 milliseconds).

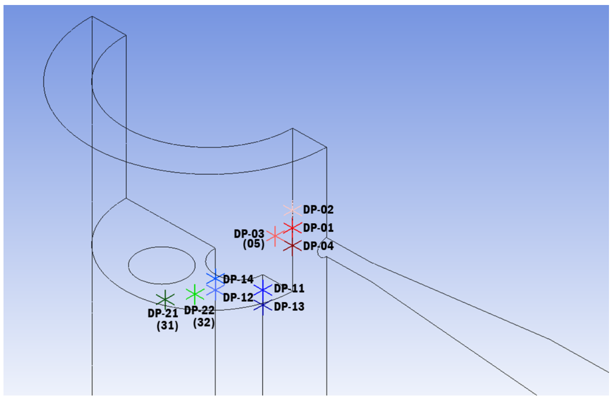


Fig. 5. Void Fraction in Experiment and CFD.

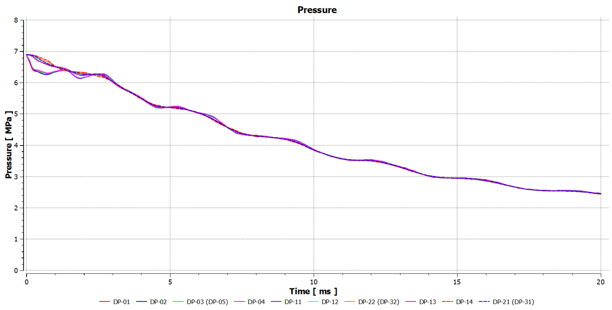


Fig. 6. Pressure Changes in CFD.

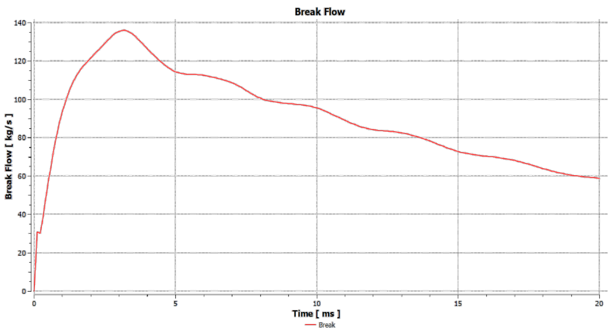


Fig. 7. Break Flow Rate in CFD.

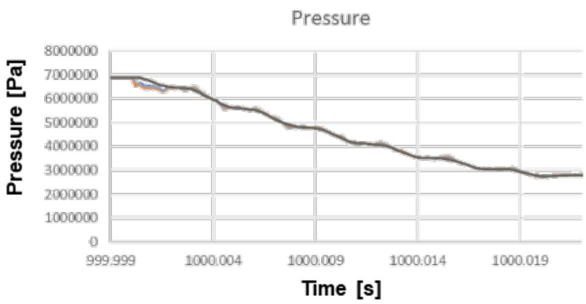


Fig. 8. Pressure Changes in RELAP5 [2].

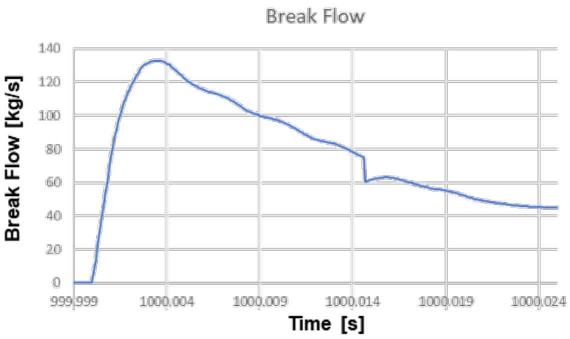


Fig. 9. Break Flow Rate in RELAP5 [2].

3. Conclusions

A three-dimensional analysis of the flow field during an accident can provide useful information for evaluating structural integrity of the component which sometimes can be neglected in a simplified analysis. In this study, the first 20 msec of a virtual break accident is simulated with ANSYS CFX 19.2. The pressure of the tank representing the pressure vessel decreases over time with pressure wave as the water jet is discharged from the tank through the break. Meanwhile, the results obtained in CFX surprisingly have a good similarity with RELAP5 results.

ACKNOWLEDGMENTS

We would like to acknowledge the technical support from ANSYS Korea, South Korea.

REFERENCES

[1] Jo, Jong Chull, et al. "Numerical prediction of transient hydraulic loads acting on PWR steam generator tubes and supports during blowdown following a feedwater line break." *Nuclear Engineering and Technology* 53.1 (2021): 322-336.

[2] Jung, Hansik. RELAP5 Code Simulation for SG FLB Experiment. 2022 Summer NuSTEPMeeting.