CFD Analysis of a Free-Surface Movement in the Safety Injection Tank using a Compressible Air Ideal Gas Model and an Incompressible Water Model

Sang-Gyu Lim*, Keuk-Jong Yu, Han-Gon Kim
Advanced Reactors Development Office, Central Research Institute, Korea Hydro & Nuclear Power Company, Ltd., 70, 1312-gil, Yuseong-daero, Yuseong-gu, Daejeon 305-343, Republic of Korea
**Corresponding author: sglim@khnp.co.kr*

1. Introduction

The Fluidic Device (FD) has been adopted in Safety Injection Tanks (SITs) as one of Advanced Design Features (ADFs) for APR1400. This device performs the control of a safety injection flow rate with a passive manner during a reflood and a refill phase of LBLOCA.

A full-scale test was performed by KAERI and the test results met the design requirement of APR1400 [2]. KHNP performs a series of CFD calculation to enhance the performance of the FD more effectively and preliminary design concept of FD+ was proposed on the basis of a series of CFD analysis [3]. The previous studies have limitations because it cannot compensates the compressibility of nitrogen gas, which is located in the top of SIT, and the free-surface movement between the pressurized nitrogen gas and the safety injection water. Theses essential phenomena determine the nitrogen release time and amounts during the whole discharging period.

This paper deals with the free-surface movement, the discharging flow rate and pressure loss characteristics in the CFD calculation, which are compensated using a compressible air ideal gas model and an incompressible water model. In addition, the CFD code capability for simulating the whole phenomena of SIT is evaluated for further CFD studies.

2. CFD Models and Boundary Conditions

CFD models, being used to simulate two-fluid and free-surface movements, are described in this section. Also, boundary conditions are presented to simulate accurate physical conditions in the SIT.

2.1 CFD Models

The source of power for the discharging of safety injection water is the pressurized nitrogen which is located in the top of SIT. After the initiation of the discharging, the compressed nitrogen is polytropic expanded and the temperature of nitrogen falls as increase of nitrogen volume in the SIT. Therefore, the nitrogen compressibility should be calculated.

The nitrogen compressibility is similar with the air ideal gas compressibility hence compressed air was used in the experiment which is performed by KAERI. Therefore, the air ideal gas model is used to simulate polytropic expansion of nitrogen in the CFX code.

A horizontal free-surface exists in a spatial geometry of SIT. The free-surface model is defined by buoyancy, hydrostatic pressure and surface tension. A homogeneous free-surface model, which assumes two phases share velocity field, is used to reduce CPU time and memory to run.

In the vortex chamber, a highly turbulent flow mixing and velocity gradient of the flow occurs during the whole discharging process. The k–ω-based Shear Stress Transport (SST) model was designed to give a highly accurate prediction of the onset and the amount of flow separation under adverse pressure gradients by the inclusion of transport effects into the formulation of the eddy-viscosity. In this calculation, therefore, SST turbulence model is applied.

2.2 Initial and Boundary Conditions

The initial SIT water level, which means the free-surface height, is set by 8.9m. Water and air volume fraction are defined using a followed STEP function.

$$WaterVF = step((8.9m - y)/1[m]) \qquad [1]$$

$$AirVF = 1 - [step((8.9m - y)/1[m])] \qquad [2]$$

Water hydrostatic pressure and air initial pressure are declared using CFX expression language (CEL) as followed equation.

$$HydroP = \rho_{water} g(8.9m - y) * WaterVF + 40bar \qquad [3]$$

Outlet boundary condition is set from 40bar to 1bar during 5 seconds, which is to compensate the characteristics of quick opening valve.

3. Calculation Results

3.1 Pressure and mass flow calculation result

The pressure of compressed air falls down due to the expansion of the air volume. During 5 seconds, CFD calculation of pressure is well predicted. After 5 seconds, pressure of calculation is over-predicted compared with experimental result as presented in figure 1.

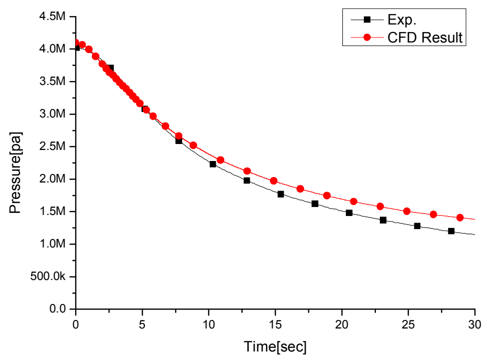

Fig. 1. Comparison of air pressure between experiment and CFD calculation

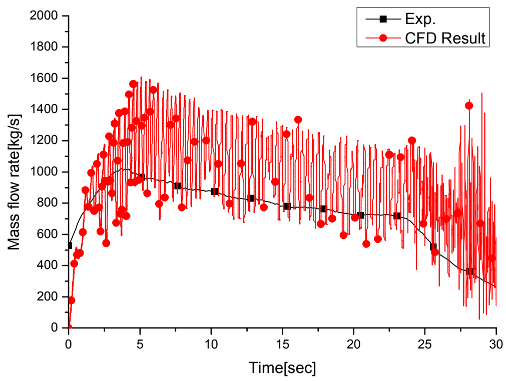

Fig. 2. Comparison of discharging flow rate between experiment and CFD calculation

The discharging flow rate of the CFD calculation shows a big oscillation and over-prediction comparing with the experimental data as shown in figure 2. The fluid velocity near the discharge nozzle is largely accelerated and a local pressure of fluid is suddenly dropped hence vaporous cavitation can occur when the local pressure drops below the vapor pressure of a liquid. In this study, cavitation effect is not considered due to a numerical complexity. Therefore, the discharge flow rate is over-predicted due to the local pressure oscillation near the throat of the nozzle.

3.2 Free-surface movement

The free-surface movement between the compressed air and the safety injection water are presented in figure 3. The free-surface movement is well predicted in the CFD calculation. In the period of turn down from high flow to low flow, the level of the stand pipe is rapidly dropped compared with the SIT level. This is due to the inertia of the liquid flow in the stand pipe. The velocity of stand pipe is faster than the velocity of SIT level therefore the stand pipe level is rapidly dropped when the SIT level drops below the stand pipe.

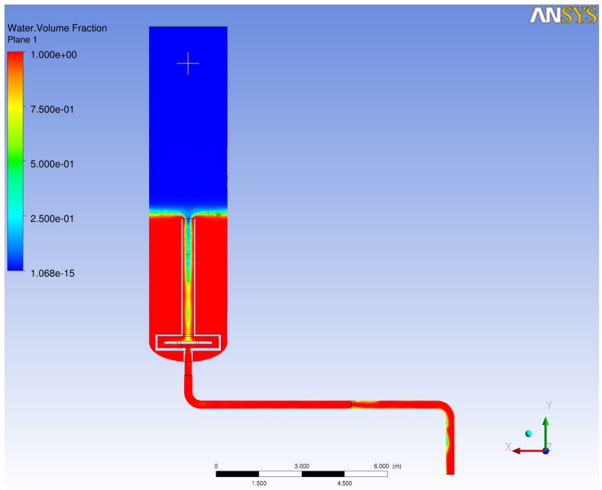

Fig. 3. Free-surface movement in the period of turning down

4. Conclusions

CFD calculation is performed using a free-surface model with an air ideal gas model and a water model. The fact that pressure of nitrogen in the experiment is complied with an ideal gas equation based is proved by the CFD calculation. Understanding of free-surface movement in the turn-down period is enhanced on the basis of the free-surface simulation.

However, there are several limitations in the CFD calculation. Local velocity and pressure near the throat of discharge nozzle are not well accurate without taking into account the vaporous cavitaion effect. Although CFD code can provide the Reyleigh-Plesset cavitation model, a numerical complexity and cost should be increased when using the cavitation model.

REFERENCES

[1] ANSYS, CFX 13.0 User Guide, ANSYS SYSTEM, 2011.
[2] Chu, I.C., Song, C.H., Cho, B.H., Park, J.K., Development of Passive Flow Controlling Safety Injection Tank for APR1400, Nuclear Engineering and Design 238, 200–206, 2008.
[3] Lim, S.G., Lee, S.H., Kim, H.G., Benchmark and Parametric Study of a Passive Flow Controller for the Development of Optimal designs using a CFD code, 240, pp. 1139-1150, 2010.
[4] Jo, J.C., Yu, S.O., Numerical analysis of unsteady flow field in the RWT for the prediction of the potential for air ingression into the ECC supply lines during the SBLOCA at the KSNPs, In: ASME PVP Conference, San Antonio, TX, 2007.