CUPID Simulation of Siphon Break Experiments

Ik Kyu Park [a*], Han Young Yoon [a], Hong Beom Park[a]
[a]*Thermal Hydraulics Safety Research Division, Korea Atomic Energy Research Institute, 111, Daedeok - daero 989 beon - gil, Yuseong - gu, Daejeon, 34057, Rep. of Korea*
[*]*Corresponding author: gosu@kaeri.re.kr*

1. Introduction

Siphon breakers are usually implemented in pool-type research reactors to maintain sufficient pool water inventory in case of loss of coolant accidents. In this study, the siphon break experiments[1] are numerically simulated using CUPID code, which is CMFD code adopting two fluid model. The siphon break test facility consists of an upper tank, main drainage pipe, siphon breaker pipe. A schematic diagram of the experimental facility is shown in Fig. 1[2]. The upper tank has about 60 ton water capacity with 4 m in depth and is made of a steel plate. The diameter of main drainage pipe is 0.4m. The height differences between the end of the siphon breaker line and the pipe break locations are 11.58 m for LOCA A and 6.58 m for LOCA B, respectively. 2.5 to 0.5 inch siphon breaker lines are connected to the horizontal part of the inverted U-shape pipe. The undershooting heights from the pool water surface are 0.33 m in the case of 2.5 inch siphon breaker lines. The end of siphon breaker line is at 3.35m from the tank bottom, and the undershooting heights means that the difference from final water level to the height of the end of siphon breaker line (3.35m).

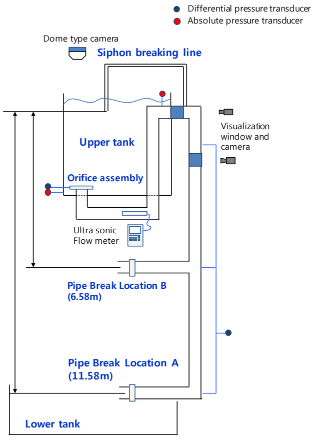

Fig. 1. Schematic diagram of the experimental facility[1]

2. Model and Results

2.1 CUPID Model

The CUPID code uses transient three-dimensional two-fluid formulation to describe the multi-dimensional two phase fluid flows. The two-fluids are the gas and the liquid. Non-condensable gas is additionally considered in the gas phase. For the mathematical closure, the interphase transfer terms of the governing equations are evaluated based upon topology map concept suggested by Tentner et al. [4].

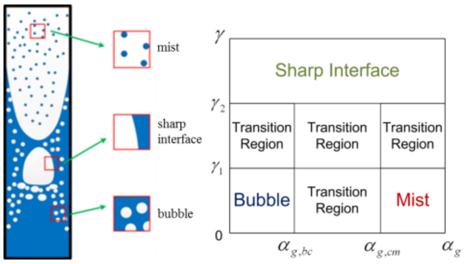

Fig. 2. Inter-phase topology configuration and topology map by void fraction and void gradient[3].

2.2 Typical Calculation

The typical calculation was conducted for the 2.5 inch siphon breaker line and the low-positioned LOCA A at -10.58 m from the end of the siphon line. The siphon experimental test section of Fig. 1 is discretized into the mesh of Fig. 3 with 328,400 hexahedral cells, 349,207 nodes, and 1,005,120 faces. The finer meshes are adopted for the inverted U-shape part of the drainage pipe to simulate siphon break phenomena precisely. The calculation domain was decomposed into 14 subdomains, and the 14 CPUs were used for parallel computing.

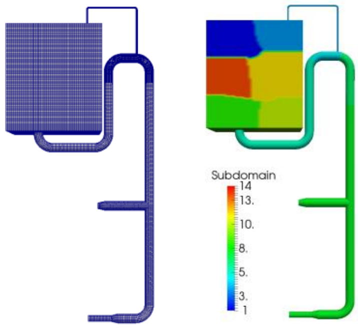

Fig. 3. Configuration of siphon mesh and decomposed domain.

The zero equation with mixing length of 0.2m and the interfacial drag model based on the topology map are adopted for this typical base calculation. The uncertainties might be included when the bubble size model is adopted in the inverted-U shape of the drainage pipe. Thus, the bubble with the diameter of the siphon breaker line was assumed to be 56 mm in the bubble flow region in the drainage pipe, and the sensitivity studies on the drag in the drainage pipe were discussed in the next section.

Fig.4 shows the water drainage by siphon and the siphon break by air pocket with the void fraction contours of the calculation domain. Initially, the calculation domain of the water tank, main drainage pipe, and siphon break was filled with the standstill water, and the tank water level was 4.0m (0 s).

The calculation was started with the event of LOCA at the end of drainage pipe, and the water was drained and tank water level was decreased by the siphon phenomena through the drainage pipe (0~20s). After the tank water level was reduced below 3.35 m, the end of the siphon breaker line was uncovered and the air started to be intaken into siphon breaker line instead of water(20s). The air intaken into siphon breaker line was delivered into the inverted U-shape partof drainage pipe and the air pocket was growing (20~30s). The continuous siphon water column was broken by the air pockets and the tank water reservoir was preserved by this siphon break (40s).

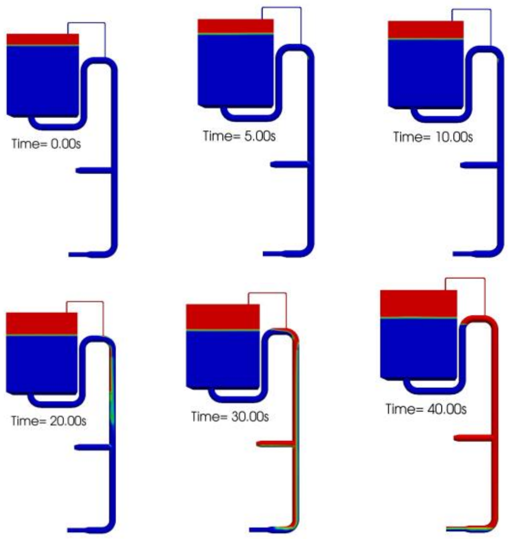

Fig. 4. Calculated water level transients(0~20s) and siphon break phenomena(20~40s)

In Fig. 5, the calculated water level of CUPID code was compared with the experimental results and the calculation results of one dimensional system analysis code, RELAP5[3]. The calculated water level of CUPID code agrees with experimental one in the decrease rate and the stop of level decrease well. It indicates that the adopted wall friction and the drag model are physically reasonable in simulating this siphon experiments. In case of RELAP5, the friction model is reasonable, but the drag model based upon the one dimensional flow regime map is not appropriate to simulate siphon phenomena.

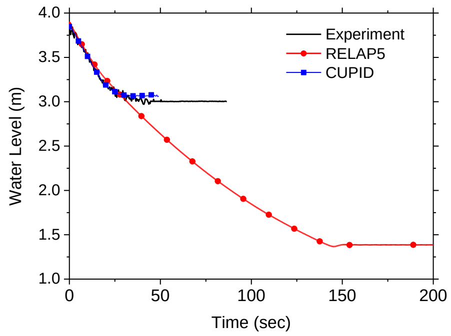

Fig. 5. Comparison of water level transients with experimental results and RELAP5 calculation[3].

The siphon phenomena can be divided into two stages: one is drainage stage (0~20s) and the other is siphon break stage (30~40s) in the Fig. 4. The water level was decreased by the siphon phenomena in the drainage stage, in which the flow rate is governed by the wall friction. In the second siphon break stage, the siphonage water column is broken by the intaken air, in which the siphon break phenomena seem to be governed by drag between water and intaken air. Thus, the sensitivity studies on the drag will be discussed in Section 2.3.

2.3 Sensitivity Study

To see the drag effect explicitly, the topology map is not applied to drainage pipe and the drag coefficient of drainage pipe is adjusted from 500 to 10000. In the Fig. 6, the effect of the drag coefficient magnitude seems to be quite important to determine the siphon break, while the effect on the LOCA flow rate is little. The LOCA flow rates are all the same for all the various drag coefficients for the first 20 seconds, when the siphon breaker line is not uncovered. This indicates that the drag is little related to the LOCA flow rate. The LOCA flow rates, however, are broke down in quite different two ways: one is the siphon break for drag coefficients of 1000, 750, 500 and the other is the continuous siphonage for the drag coefficients of 2500, 5000, 10000. In the case of large drag coefficients, the entrained air from siphon breaker line into drainage pipe is dragged out by the water and cannot form the air pocket while the air pocket grows and the siphon is broken like Fig. 5 in the case of small drag coefficients.

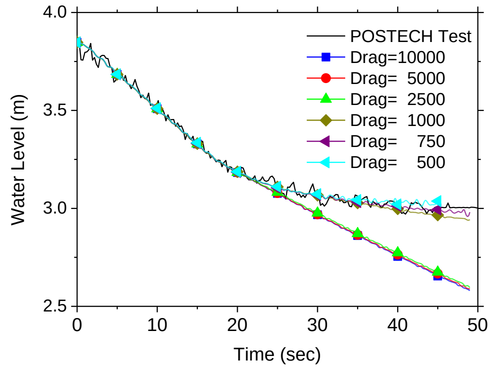

Fig. 6. Sensitivity study on the drag coefficient in the drainage pipe.

Thus, the sensitivity study on the drag coefficients in the bubbly region of drainage pipe was conducted and the results are presented in Fig. 7. The effect of the drag coefficient magnitude in the bubbly topology is revealed to be neglected in the view of LOCA flow rate and the siphon break up to drag coefficient of 5000. This is quite different from the results of the sensitivity study on the drag coefficients of drainage pipe in case of topology map free. It is concluded through these two kinds of approaches to the siphon break that the drag model according to the topology map seems to be the key to simulate the siphon break properly, though the reliable bubble diameter model is needed still.

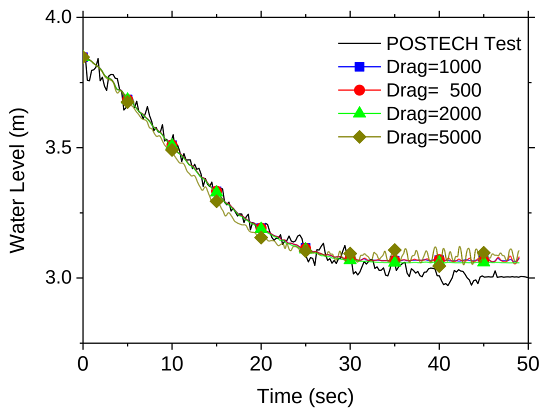

Fig. 7. Sensitivity study on the drag by drag coefficient in the bubbly region of drainage pipe.

3. Conclusions

In this study, POSTECH experiments for the siphon break are simulated using CUPID. The typical case calculation for 2.5 inch siphon breaker line and low-positioned LOCA agree with experimental data well in LOCA flow rate, siphon break time and siphon undershooting. Sensitivity study indicates that the CUPID code can be applied to predict the siphon break phenomena if the drag model to give proper value in reasonable range. After the further study such as sensitivity calculations on the friction, the CUPID code can be used as a 3-dimensional safety analysis tool for pool-type reactor LOCA.

ACKNOWLEGEMENTS

This work was supported by the National Research Foundation of Korea (NRF) and the Korea Foundation of Nuclear Safety (KoFONS) grant funded by the Korean government (MSIP & NSSC)(Nuclear Research and Development Program: 2017M2A8A4015005, Nuclear Safety Research Center Program: 1305011).

REFERENCES

[1] S. H. Kang et al., Experimental study of siphon breaking phenomenon in the real-scaled research reactor pool, Nuclear Engineering and Design, 255, 28-37, 2013.
[2] S. H. Kang et al., Investigation of effects of enlarged pipe rupture size and air penetration timing in real-scale experiment of siphon breaker, Nuclear Engineering and Technology, 46, No. 6, 2014.
[3] J. H. Chun et al., RELPA5/MOD3.3 Simulation of Siphon Break Experiments, NURETH-529, Pisa, Italy, May 12-15, 2013.
[4] A. Tentner et al., Computational Fluid Dynamics Modeling Of Two-Phase Flow Topologies In A Boiling Water Reactor Fuel Assembly, ICONE16, Orlando, Florida, USA , May 11-15, 2008.