A Numerical Study of Air Entrainment by the Pipe Rupture Size for Downward Flow Research Reactor

Minkyu Jung [a*], Ki-Jung Park [a], Kyoungwoo Seo [a], and Seong-Hoon Kim [a]
[a]*Korea Atomic Energy Research Institute, Daejeon 34057, Korea*
[*]*Corresponding author: minkyujung@kaeri.re.kr*

1. Introduction

This paper presents the numerical studies of air-water two fluid behaviors in cooling system of RR (research reactor). Normally, the cooling system could be designed as upward or downward flow in accordance with the purpose of RR utilization. For the downward flow cooling system, negative pressure distribution could be occurred at the highest pipe position of reactor core outlet. It means that air entrainment could be happened at the pipe rupture accident, because the pressure inside the pipe is lower than atmospheric pressure. This phenomenon is completely different from a LOCA (loss of coolant accident).

Many research reactors are open-pool type, and the pressure distribution of the cooling system could be computed by the pool water head and pressure difference in reactor core. Besides, pool outlet pipes are placed at higher position to guarantee the pool water inventory at LOCA. For these reasons, the negative pressure could be observed at the highest pipe position. Thus, the air could be flowed into the highest position of cooling system at the pipe rupture accident. To analyze the flow phenomena by the air entrainment and calculate the air flow rate, the experimental equipment is designed. In the present study, prior to the experimental study of air entrainment, numerical simulations are conducted by the pipe rupture size.

2. Geometric configuration and numerical methods

2.1 Geometric configuration and numerical meshes

For the present study, to simulate the air entrainment phenomena efficiently, only reactor outlet pipes are configured without any pumps, heat exchanger, and decay tank as shown in Fig. 1. The pipe diameter is 8 inches and the height for air entrainment position is 4 meters. Among the several rupture positions, the top position is considered for the air inlet.

Only hexahedral meshes are adopted by splitting to 63 blocks to conduct more accurate flow simulation. Total 3.4 million meshes are utilized for the computation.

2.2 Numerical methods

A commercial computational fluid dynamics (CFD) software, ANSYS Fluent is utilized for the calculation [1]. The fluid motion is modeled by incompressible Reynolds-averaged Navier-Stokes equations. The numerical domain is discretized using cell-centered finite volume method. To conduct the air-water two fluid simulation, VOF (volume of fluid) model is adopted. The realizable k-epsilon two-equation model is utilized to calculate the turbulent eddy viscosity with enhanced wall treatment. To enhance the spatial accuracy, second order upwind method is used for spatial discretization with Green-Gauss cell based gradient.

Velocity inlet and pressure outlet boundary condition is applied at inlet and outlet boundary, respectively. Initially, the flow simulation is conducted for the steady state with the wall condition at air inlet boundary. After changing the air inlet boundary to pressure inlet, the transient simulation is carried out.

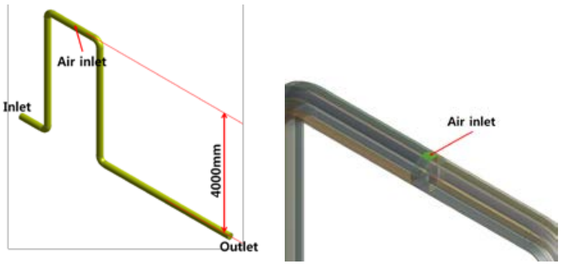
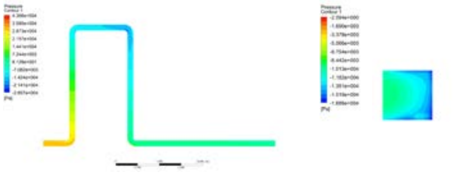


Fig. 1 Geometric configuration of downward flow pipe for air entrainment flow simulation

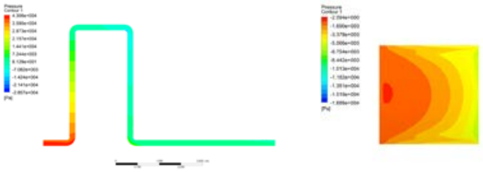
3. Result and Discussion

Prior to the transient calculation according to the pipe rupture size, the steady state simulation is conducted until the residuals are lower than 10^{-6}. Inlet velocity is 2.5 m/s and outlet gauge pressure is 0 kPa. All cases for the rupture sizes have identical pressure distribution, and the pressure difference between air inlet boundary and atmosphere is about 37 kPa.

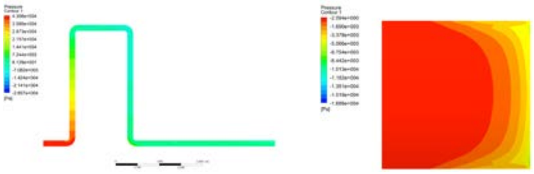
From the steady state solution, the transient simulations are conducted to advance the flow variables with 0.001 seconds time step and 20 times sub-iterations. The proportion of pipe rupture area to pipe cross-section area is 1.28%, 5.09%, 11.5%. Figure 2 shows the gauge pressure contours for three pipe rupture size ratios at center plane and rupture area. It is observed that pressure at the front and the rear of rupture position is increased by exposing the rupture area to air. It is also shown that increased pressure is also larger by the increment of the rupture size. These pressure behaviors are well-observed at rupture area. And, it is found that the pressure difference between area ratio 1.25% and 5.09% is pretty larger than that between area ratio 5.09% and 11.5%.



a) Area ratio : 1.25%

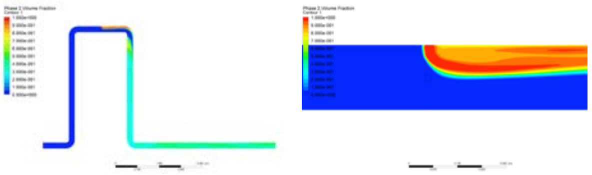


b) Area ratio : 5.09%

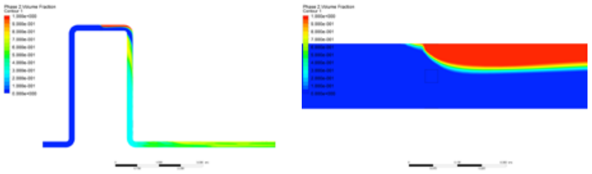


c) Area ratio : 11.5%

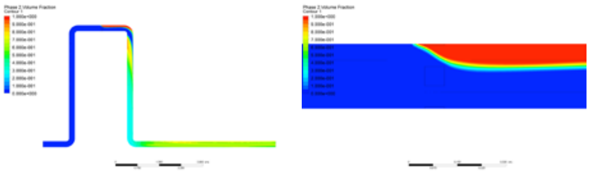
Fig. 2 Pressure contour at center plane and rupture area of the pipe for three different pipe rupture size ratios; a) 1.25%, b) 5.09%, c) 11.5%



a) Area ratio : 1.25%



b) Area ratio : 5.09%



c) Area ratio : 11.5%

Fig. 3 Air volume fraction contour at center plane and near the rupture position for three different pipe rupture size ratios; a) 1.25%, b) 5.09%, c) 11.5%

Air volume fractions are compared by the different pipe rupture size in Fig. 3. It is observed that air is flowed into the pipe from the top position and exited to outlet boundary in all cases. It can be shown that overall air volume fractions are similar except for local air volume fraction magnitude near the air inlet position. The lower air volume fraction is observed at area ratio 1.25% case because the air inflow amount is less than other cases due to the small rupture area. However, it cannot be found the much difference between area ratio 5.09% and 11.5%. This flow characteristic is similar to the pressure distribution previously explained in Fig. 2.

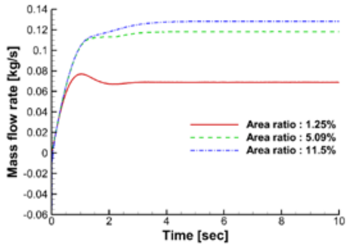


Fig. 4 Mass flow rate variation at rupture area for three different area ratio cases; 1.25%, 5.09%, and 11.5%

The comparison of the mass flow rate at the pipe rupture area over time is shown in Fig. 4. In all cases, air entrainment amount is increased after the pipe rupture, then the mass flow rate is converged after 3 seconds. The converged mass flow rate is 0.0689 kg/s, 0.118 kg/s, and, 0.128 kg/s for three cases, respectively. As previously explained at pressure distribution and air volume fraction, difference of the mass flow rate between area ratio 5.09% and 11.5% cases is less than that between 1.25% and 5.09%. It can be inferred that there's not much difference in the flow characteristics in spite of the rupture size increase because a decent portion of pressure is increased at area ratio 5.09% case. Similar results were reported in a siphon break experimental study, which calculated the air mass flow rate in a siphon line by the atmospheric pressure [2]. The negative mass flow rate is observed for area ratio 5.09% and 11.5% cases at the beginning. It means that small amount of water is discharged at the pipe rupture area.

4. Conclusions and Future works

This study was performed to analyze the flow phenomena on the air entrainment and calculate the air flow rate according to the pipe rupture size. All simulations were conducted for the cases with negative pressure at the highest pipe position. As the rupture size is increased, the pressure difference between inside pipe and atmosphere is decreased and air entrainment amount is increased. It is found that the air entrainment increment is decreased for the largest rupture size, because the pressure at the highest pipe is substantially recovered at the medium rupture size in a given pressure difference. From this study, another parameters, such as pressure difference, rupture direction, fluid velocity, and so on, will be also tested. And, this study will be applied to design the experimental equipment for air entrainment.

ACKNOWLEDGMENT

This work has been conducted as a part of the Development of Research Reactor Technology project sponsored by Ministry of Science and ICT of Korean government.

REFERENCES

[1] ANSYS FLUENT theory guide, ANSYS Inc., 2017.
[2] KAERI/CM-1902/2013, Experimental Study of Siphon Breaker.