Transactions of the Korean Nuclear Society Spring Meeting
Jeju, Korea, May 18-19, 2023

Assessment of Helium Bubbling Effect in terms of Natural Circulation Performance by Using Experiment and CFD

Wonjun Choi[a], Jae Hyung Park[a], Juhyeong Lee[a], Jihun Lim[a], Yunsik Cho[a], Sangtae Kim[a], Yonghee Kim[c], Youngsoo Yoon[d] and Sung Joong Kim*[a,b]
[a]*Department of Nuclear Engineering, Hanyang University*
222 Wangsimni-ro, Seongdong-gu, Seoul 04763, Republic of Korea
[b]*Institute of Nano Science and Technology, Hanyang University*
222 Wangsimni-ro, Seongdong-gu, Seoul 04763, Republic of Korea
[c]*Department of Nuclear and Quantum Engineering, Korea Advanced Institute of Science and Technology*
291 Daehak-ro, Yuseong-gu, Daejeon 34141, Republic of Korea
[d]*Department of Materials Science and Engineering, Gachon University*
1342 Seongnam-daero, Sujeong-gu, Seongnam-si, Gyeonggi-do 13120, Republic of Korea
** Corresponding author: sungjkim@hanyang.ac.kr*

1. Introduction

To develop the innovative systems for the molten salt reactor (MSR), the Republic of Korea established the i-SAFE-MSR research center in 2021. It proposed an advanced MSR, the passive molten salt fast reactor (PMFR), which is operated through natural circulation without reactor coolant pumps (RCPs). The operation of natural circulation can prevent undesirable transitions such as corrosion of pumps etc. In addition, PMFR has been being designed as concepts of severe-accident free, low radioactive waste, and long-lifetime operation up to 40 years etc. Figure 1 shows 3D schematic of PMFR.

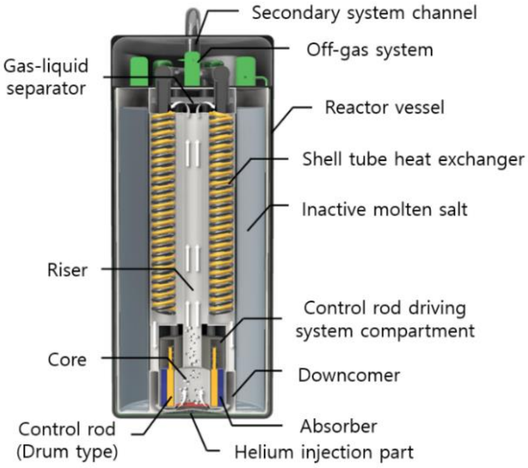

Fig. 1. 3D schematic of PMFR

In the PMFR, a major issue related with fission products exists because the molten salt mixture including fission products circulates in the primary system. In particular, the presence of non-soluble fission products in the primary loop of PMFR can affect reactor operation. For example, if the non-soluble fission products stick to materials, decreasing of heat transfer efficiency and local corrosion can be accelerated. To address the issue, helium bubbling system which has been used frequently in the MSR was adopted in PMFR [1].

Helium bubbling system is to remove non-soluble fission products by injecting the helium bubbles in the primary system. When the helium gas is injected through inlets located at the lower part of system, the non-soluble fission products are attached to the surface of helium bubbles. The helium bubbles combined non-soluble fission products rise toward the free surface. When helium bubbles attaching the non-soluble fission products reach the free surface, the bubbles burst and the fission products can be collected in a collection device of fission products located at the upper part of system.

The helium bubbling system can also improve the natural circulation performance of PMFR system which is operated without RCPs. When the helium bubbles were injected, the working fluid can receive the buoyant force from the density differences between helium bubbles and working fluid (molten salt). The buoyancy can play a role of driving force on working fluid's circulation. Furthermore, enhanced natural circulation performance of working fluid can increase the available thermal output when the operation conditions are identical [2].

To confirm the helium bubbling effect in terms of natural circulation performance, appropriate experiment should be conducted. However, it is difficult to directly assess the effect of helium bubbling system in PMFR experimentally. Because the size of PMFR is large and PMFR includes radioactive materials such as uranium etc. Therefore, a scaled-down two-phase molten salt natural circulation experiment facility needs to be designed to simulate the helium bubbling effect. Based on the two-phase molten salt natural circulation experiment, PMFR analysis code platform needs to be developed to analyze the helium bubbling effect in the real condition of PMFR.

In this study, two-phase adiabatic natural circulation experiment was preliminary conducted before performing the two-phase molten salt natural circulation experiment. Because the two-phase molten salt natural circulation experiment required insulators which disturb the visualization of helium bubbles. Furthermore, it is more complicated to design the two-phase molten salt natural circulation experiment. Thus, the objectives of this research are to confirm the helium bubbling effect by two-phase adiabatic natural circulation experiment and compare the results between experiment and computational fluid dynamics (CFD) code. Based on the results of two-phase adiabatic natural circulation experiment, accuracy of CFD code was assessed.

2. Two-phase adiabatic experiment

Figures 2 (a) and (b) show a schematic of two-phase adiabatic experimental loop and actual experimental facilities, respectively. The experimental facilities consist of a riser, an upper pool, a downcomer, a buffer tank, a bottom line and inlet parts. Deionized (DI) water was selected as the simulant fluid based on its similarity of the kinematic viscosity (0.8~1.0 mm^2/s) under room temperature with the molten salt mixture under the operating condition of the PMFR such as NaCl-UCl$_3$ (0.65~1.3 mm^2/s) or KCl-UCl$_3$-UCl$_4$ (0.63~1.05 mm^2/s). The amount of helium gas injected into the system was controlled by using the mass flow controller from 1 to 10 L/min (lpm).

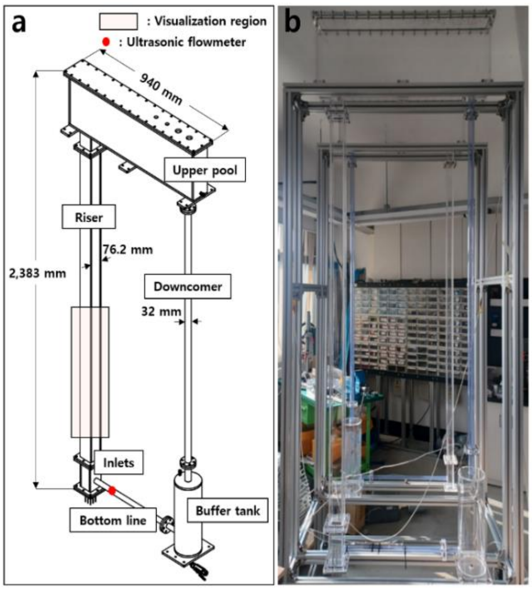

Fig. 2. (a) A schematic of experimental facility, and (b) two-phase helium bubbling experimental loop

When the helium bubbles are injected by upward direction through inlets located at the lower part, DI water receives the buoyancy from helium bubbles. As a result, DI water in the riser moves upward with helium bubbles. In the upper pool, the helium gas injected through inlets is released and the DI water which rises up along the riser goes down through downcomer. The DI water passes through the buffer tank and flows along the bottom line. The buffer tank can relieve the fluctuation of pressure and remove the helium bubbles near the buffer tank and bottom line. The DI water flowing along the bottom line moves again to riser and the DI water can circulate the entire system due to driving force induced by helium bubbles.

As major parameters which should be observed in the experiment, the water velocity and void fraction were selected. The water velocity is a parameter which can be used to confirm the degree of natural circulation. In addition, the void fraction is a significant variable to affect the flow distribution in two-phase flow. The water velocity was measured from ultrasonic flowmeter installed at the bottom line as shown in Fig. 2 (a). The void fraction was calculated from empirical correlation as shown in Eq. (1) which proposed by Fan et al. [3]. In Eq. (1), U_T, $U_{g,s}$, $U_{l,s}$, $U_{s,s}$ are terminal rise velocity of a single bubble, superficial velocity of gas-phase, liquid-phase and solid-phase, respectively. Terminal rise velocity (U_T) was assumed as 0.35 m/s referring to the previous research.

$$\varepsilon_g = \frac{U_{g,s}}{U_T + 1.1\left(U_{g,s} + U_{l,s} + U_{s,s}\right)} \tag{1}$$

Visualization of flow was also performed with high-speed camera to confirm the change of flow pattern according to the helium injection rate. Figure 3 shows the behavior of a two-phase flow captured by a high-speed camera according to helium injection rate of 1, 5, 9 lpm. It was confirmed that according to the helium injection rate, the flow distribution of working fluid and the performance of natural circulation can be changed as shown in Fig. 3. All experiments were conducted at the condition of normal pressure and 20 °C temperature.

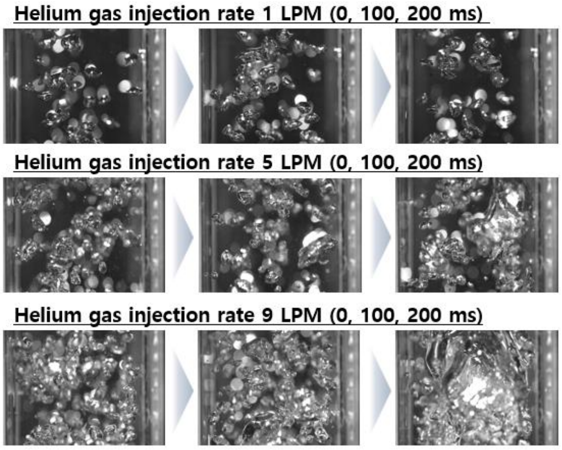

Fig. 3. Visualization of a two-phase flow (1, 5, 9 lpm)

3. Numerical calculation

3.1. Geometry and mesh

To calculate major parameters such as water velocity and void fraction by using the numerical method, it is required to make geometry and mesh of the analysis domain. The geometry and mesh were made by SALOME-9.7.0, an open-source computer-aided design (CAD) software as shown in Fig. 4. The geometry of analysis domain consisted of a riser, an upper pool, a downcomer, a buffer tank, a bottom line and inlets as same with experimental facilities.

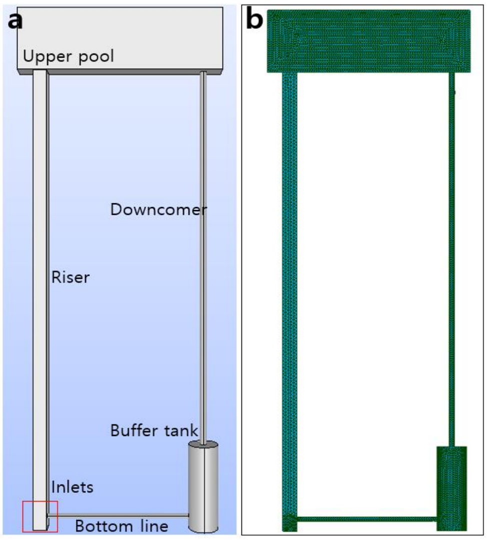

Fig. 4. (a) Geometry, and (b) mesh of the analysis domain

The mesh was fabricated by using the main algorithm as NETGEN 1D-2D-3D based on tetrahedral mesh. The mesh quality of analysis domain was assessed through the investigation of mesh sensitivity. To conduct the assessment of mesh sensitivity, the mesh cases were classified with three groups: coarse mesh, moderate mesh and fine mesh. The number of cell meshes corresponding to each case was 215,533 in coarse mesh, 296,930 in moderate mesh and 426,128 in fine mesh.

To investigate the mesh sensitivity, a reference case where the helium injection rate is assumed as 1 lpm (~ 1.667×10^{-5} m^3/s) was selected. The assessment of mesh sensitivity was performed by comparing each value of water velocity and void fraction obtained from three mesh groups about the reference case. The parameters such as the void fraction and water velocity were little changed except for slight fluctuations after 15 seconds from starting the simulation. In other words, 15 seconds was decided as time taken to reach the steady-state. Thus, an assessment of mesh sensitivity was conducted by utilizing the values of water velocity and void fraction after 15 seconds from starting the simulation.

According to the assessment of mesh sensitivity, the differences of water velocity and void fraction between coarse mesh and fine mesh in the steady-state were less than 2.73 % and 1.62 %, respectively as shown in Fig. 5. In other words, differences of values among three mesh groups were little. Even if it is expected that the results obtained by fine mesh are more accurate, the coarse mesh was finally selected for calculation by considering computational time.

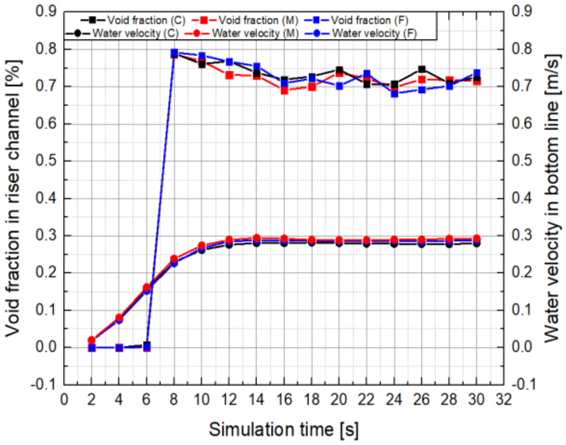

Fig. 5. Void fraction in the riser channel and water velocity in the bottom line according to mesh quality (C: coarse mesh, M: moderate mesh, F: fine mesh)

3.2. Parameters of OpenFOAM simulation

Based on the coarse mesh selected from mesh sensitivity assessment, the numerical calculation was conducted by using OpenFOAM, one of open-source CFD codes. In OpenFOAM, various solvers and utilities for calculation of thermal hydraulic phenomena exist. To analyze two-phase flow including water and helium bubbles, multiphaseEulerFoam solver which can be utilized for the system including multi-phase fluids was selected. The drag flux model was selected as Schiller-Naumann model which has been used generally for calculation of two-phase drag force. The main parameters used in the helium bubbling calculation are represented in Table 1.

Table 1. Major parameters for OpenFOAM calculation

Parameters	Values
Simulation	
Solver	multiphaseEulerFoam
Turbulence model	laminar
Time step	0.0004 (1 lpm) ~ 0.00002 (10 lpm)
Iteration and discretization	
Iterative solver	PIMPLE (Pressure -Velocity coupling algorithm)
Smoother	symGaussSeidel
Time term	Euler
Gradient term	Gauss linear
Drag flux model	Schiller-Naumann segregated
Velocity boundary condition	
Inlet	Fixed value (e.g. 1 lpm ~ 0.7442 m/s)
Outlet	pressureInletOutletVelocity
Walls	No slip condition

4. Results and Discussion

Based on the experimental and numerical results, the comparison of major parameters such as water velocity and void fraction was conducted. Figures 6 and 7 show the water velocity in a bottom line and void fraction in a riser channel obtained by experiment and OpenFOAM, respectively. The location where the water velocity was extracted in OpenFOAM was set to be the same as the location where the ultrasonic flowmeter was installed in the experiment.

The water velocity in a bottom line obtained by experiment and OpenFOAM increased from 0.338 to 0.904 m/s and from 0.278 to 0.842 m/s as the helium gas injection rate increased from 1 to 10 lpm, respectively. In other words, the enhancement of natural circulation induced by helium gas was confirmed both in the experiment and OpenFOAM. However, the gradient of water velocity decreased due to the two-phase frictional pressure drop as the helium gas injection rate increases.

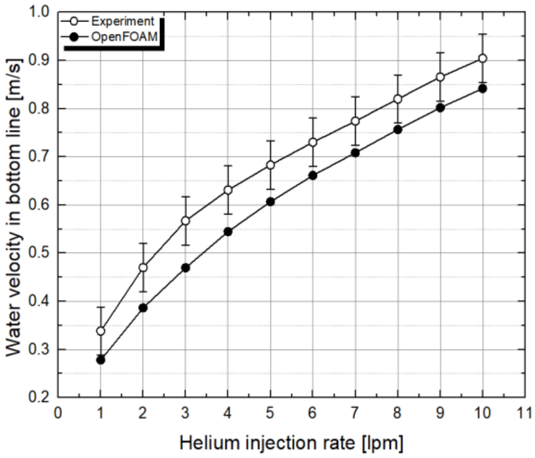

Fig. 6. Comparison the water velocity in a bottom line between experiment and OpenFOAM

The relative percentage error of water velocity between experiment and OpenFOAM ranged in 6.93~17.70 %. The tendency of water velocity between experiment and OpenFOAM was presented similarly with each other. However, it was confirmed that the water velocity calculated by using OpenFOAM was slightly underestimated compared to the experimental values. The cause of discrepancy is estimated because consideration of helium bubble size's variation under pressure in OpenFOAM was not reflected properly. According to experiment, helium bubbles expanded due to decrease of hydrostatic pressure as the helium bubbles move upward. As opposed to the experiment, it seemed that the expansion of helium bubbles according to moving upward in the riser was not considered appropriately in OpenFOAM calculation. This resulted in the underestimation of the driving force induced by the density differences between the helium bubbles and water. As a result, it is predicted that the water velocity calculated by using OpenFOAM would also be underestimated compared to the experimental data.

The average void fraction in a riser channel obtained by experiment and OpenFOAM increased from 1.19 to 9.45 % and from 0.72 to 6.17 %, respectively as helium gas injection rate increased from 1 to 10 lpm. The relative percentage error of void fraction between experiment and OpenFOAM ranged in 34.72~39.28 %. The void fraction calculated from OpenFOAM was also underestimated compared to experimental values as shown in Fig. 7.

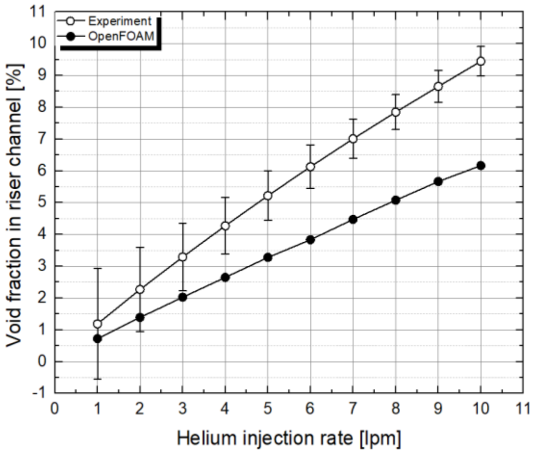

Fig. 7. Comparison the void fraction in a riser channel between experiment and OpenFOAM

It is predicted that the major reason for the underestimation of void fraction in OpenFOAM was related to the population balance equation. The population balance equation, which can simulate bubble interactions such as bubble coalescence and bubble break-up was not included during the OpenFOAM calculation. Thus, the complex phenomenon such as bubble dynamics was not considered when the void fraction was calculated in OpenFOAM and because of this, it is assessed that the errors of void fraction between experiment and OpenFOAM occurred.

It is estimated that the additional reason for underestimation of void fraction is owing to low mesh quality of inlets. Because size of inlets is too small as inner diameter 1.78 mm, if the overall mesh size is adjusted to inlet mesh size, the number of cell meshes on entire domain significantly increases. The large number of meshes can cause a huge computational time. Thus, the mesh quality of inlets was intentionally lowered, and as a result, it is assessed that the helium gas injection rate would be applied differently from user input values. Although it was confirmed that the number of entire cell meshes did not affect the result significantly according to investigation of mesh sensitivity, the inlet mesh quality can affect the result.

The drag flux model can be also proposed as another reason to cause the error of void fraction between experiment and OpenFOAM. Xiao et al. proposed that the drag force between two phases can affect the void fraction calculation and to resolve the underestimation of void fraction in CFD, it is required that the appropriate correction factor is included [4].

Consequently, to achieve accurate simulation of two-phase flow, it is required that the reflecting of population balance equation, modification of the inlet mesh quality, and selection of appropriate drag flux model. Based on the experimental data, the OpenFOAM code will be adjusted to enable accurate calculation of two-phase flow.

5. Conclusion

This study aimed to investigate the impact of helium bubbling on two-phase natural circulation through experimental and numerical analysis. The behavior of helium bubbles was visualized and evaluated through a two-phase adiabatic natural circulation experiment. Additionally, because OpenFOAM will be used to develop the PMFR analysis code platform, the accuracy of the results obtained by OpenFOAM was validated against experimental data to identify needed improvement. The major findings of this study can be summarized as follows:

- ✓ According to the experiment and OpenFOAM, as the helium gas injection rate increased from 1 to 10 lpm, water velocity in the bottom line increased from 0.338 to 0.904 m/s and from 0.278 to 0.842 m/s, respectively.
- ✓ According to the experiment and OpenFOAM, as the helium gas injection rate increased from 1 to 10 lpm, the void fraction in the riser channel increased from 1.19 to 9.45 % and from 0.72 to 6.17 %, respectively.
- ✓ The relative percentage error of water velocity and void fraction between experiment and OpenFOAM showed 6~17 % and 34~39 %, respectively
- ✓ The error of the water velocity and the void fraction between experiment and OpenFOAM could be due to the drag model and inlet mesh qualities etc.

Based on the two-phase adiabatic natural circulation analysis, two-phase molten salt natural circulation experiment will be performed. Furthermore, the additional analysis including sensitivity assessment of interfacial drag flux model will be conducted to improve the numerical analysis code.

Acknowledgments

This research was supported by the National Research Foundation of Korea (NRF) and funded by the ministry of Science, ICT, and Future Planning, Republic of Korea (grant numbers NRF-2021M2D2A2076382).

REFERENCES

[1] Journée, Dirkjan. "Helium bubbling in a molten salt fast reactor." Delft: TU delft: 7-12, (2014).

[2] LIM, J., et al. Preliminary Analysis of the Effect of the Gas Injection on Natural circulation for Molten Salt Reactor Type Small Modular Reactor System Operated without a Pump. In: Transactions of the Korean Nuclear Society Virtual Autumn Meeting. p. 21-22, (2021).

[3] Livingston, A. G., and S. F. Zhang. "Hydrodynamic behaviour of three-phase (gas—liquid—solid) airlift reactors." Chemical Engineering Science 48.9: 1641-1654, (1993).

[4] Liang, Xiao-Fei, et al. "CFD-PBM approach with modified drag model for the gas–liquid flow in a bubble column." Chemical Engineering Research and Design 112: 88-102, (2016).