CFD Simulation of a Bubble Mixing in a Square Duct

Tae-Soon Kwon and **Bub-Dong Chung**

* Korea Atomic Energy Research Institute, Yuseong-Ku Daejeon, Korea
E-mail : tskwon@kaeri.re.kr

1. Introduction

A feasibility and applicability of a CFD two-phase model for an air-water mixing in a square duct has been studied using the CFX code. Since the interfacial drag model of system code, such as RELAP5 has been developed based on the 1-D drift velocity, the CFD simulation would be a useful method to validate for a 2-D downcomer boiling model. To simulate the downcomer boiling, an air injection to a one-sided wall was considered to validate the bubble rise velocity for a downcomer flow recirculation. The bubble raising pattern and flow distribution were compared with experimental results. For this simulation, a fine mesh of tetrahedron and the MUSIG model were applied.

2. Experimental Condition

The air-water test section has a dimension of 250x250 mm in cross section, and 4.3 m in length. The schematic of the test facility and the location of the velocity measurement are displayed in Fig.1. The bubble generator was installed at the bottom and one-sided wall of the test section. The air injection holes of the bubble generator are located at a single line. The total number of holes is 203, and the diameter of each hole is 0.97mm. The non-dimensional air injection velocity in the test section, j_g^*, varies from 0 to 1.1. Test conditions are summarized in Table 1.

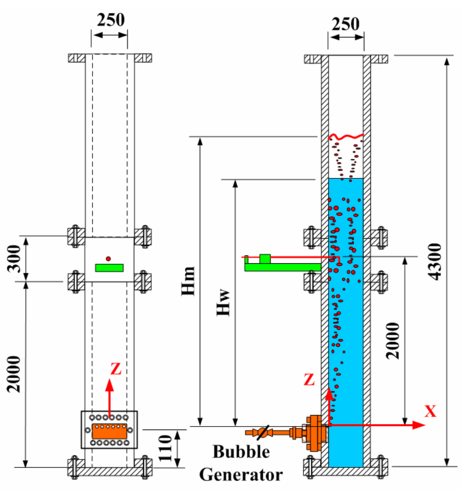


Fig. 1 Slab test facility

Table 1. Summary of the test conditions

Item	Value
Air injection, g/sec	0.12~11.3
Pool Pressure, atm	1
Pool Temperature, ^{o}C	15
Initial Water Level(H_W),m	~260

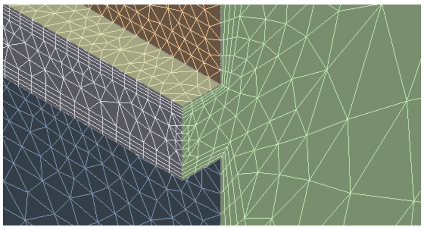


Fig. 2 CFX mesh at air INLET

3. Numerical Model

A symmetric cross section having a half width is adopted for the 3-dimensional model of the test section. Total number of 831,800 elements and 690,000 tetrahedron meshes were applied for the internal flow field of the test section. The thermal mixing and mass transfer between an air bubble and water was not considered in this simulation. The GRACE model was applied for a drag model of air bubble. For the bubble motion and breakup, the MUSIG model was applied. A free slip condition was applied for an air-bubble while a no-slip condition was applied for the water at the walls. The degassing boundary condition was applied at the top of the test section to prevent bubble accumulation.

4. RESULTS

4.1 Experimental Velocity Range

The bubble velocity was measured by a two probes method[1]. The sensor is located at 2m above the top of the bubble generator, and traversed in the horizontal direction at the center of the test section. The averaged cross sectional bubble velocity is defined as Eq. (1).

$$Bubble\ Velocity = \frac{1}{N}\sum_{i=1}^{N} U_Z(X=i) \qquad (1)$$

The non-dimensional air velocity, j_g^*, is defined as Eq. (2).

$$j_g^* = \frac{j_g}{\left[\frac{\sigma g(\rho_f - \rho_g)}{\rho_f^2}\right]^{\frac{1}{4}}} \qquad (2)$$

As shown in Fig.3, the bubble rise velocity is increased very rapidly with j_g^* in the low range below 0.4, and then reaches a finite value closed to 1.0.

.

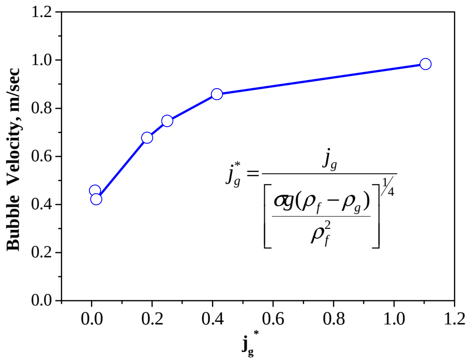
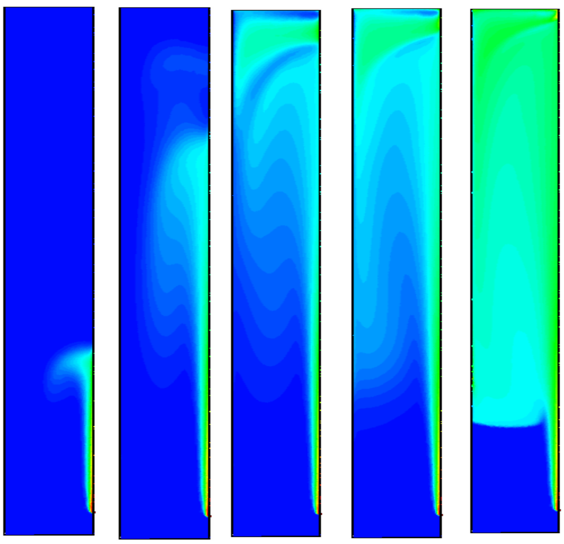


Fig.3 Bubble velocity

4.2 Numerical Visualization

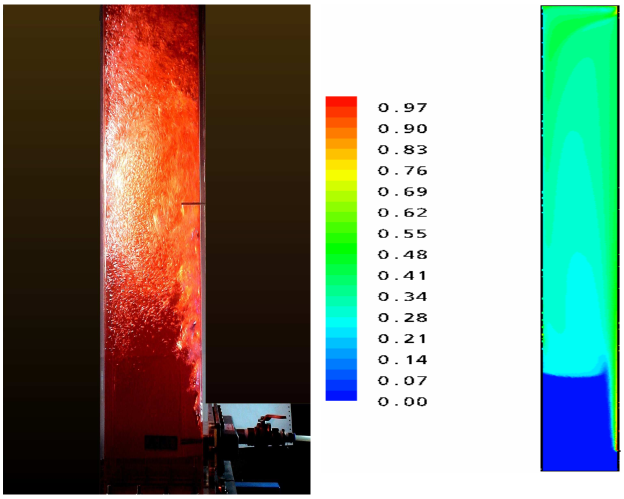
The developing bubble distribution pattern is well displayed in Fig.4. In the bottom right hand side is the bubble generator. Due to the single side wall injection, the bubble is only raised at the right hand side wall. The recirculational bubble pattern appears at the left hand side of the test section. This large recirculational flow pattern is induced by the raised bubbles which are injected through the bubble generator at the right hand side.



1/5T 2/5T 3/5T 4/5T 5/5T

Fig.4 Propagation shape of an air bubble

The visualization results are displayed in Fig.5. Fig. 5(b) shows the local void fraction. The bubble distribution of the CFX simulation has a very similar shape when compared to the experimental result.



(a) Experiment (b) CFX Visualization

Fig.5 Bubble injection shape, m_a=2.56 g/sec

5. Conclusions

The feasibility of a CFX application for a two-phase model was performed for an air-water pool mixing with the condition of a single side wall bubble injection. The numerical visualization result has a similar shape when compared to the experimental. It would be possible to apply a CFX model to validate a drift velocity correlation such as multiplication factors. But, it is also needed to evaluate a bubble break up and its drag for a realistic simulation.

ACKNOWLEDGMENTS

This research has been performed as a part of the nuclear R&D program supported by the Ministry of Science and Technology (MOST) of the Korean government.

REFERENCES

1. D.J. Euh,B. J. Yun and C. H. Song, Numerical simulation of an improved five-sensor probe method for local interfacial area concentration measurement, *Nuclear Engineering and Design*, Vol. 234, pp.99-116, 2004
2. N. Zuber and J. Findlay, Average Volumetric Concentrations in two-phase flow systems, *J. Heat Transfer*, vol. 87, pp. 453-568,1965.
3. I. Kataoka and M. Ishii, Drift flux model for large diameter pipe and new correlation for pool void fraction, *In'l J of Heat Mass Transfer*, Vol. 30, pp.1927-1939, 1987.