# CFD Simulation of Subcooled Boiling Flow

Sung Gil Shin, Soyoung Lee, Jeong Ik Lee*
*Dept. Nuclear & Quantum Eng., KAIST, 291, Daehak-ro, Yuseong-gu, Daejeon, 34141, Republic of Korea*
**Corresponding author: jeongiklee@kaist.ac.kr*

## 1. Introduction

Assessing nuclear power plant's safety under various severe accident scenarios is becoming a new global norm of nuclear safety. Complex two-phase flow phenomena occur and govern the consequences under a severe accident, a conventional one-dimensional system analysis code used for design basis accident analysis is facing a new challenge. A Computational Fluid Dynamics (CFD) approach with full three-dimensional (3D) capability to model complex two-phase flow will become very important for predicting severe accident consequences, which will lead to the development of better prevention and mitigation strategies for severe accidents in a nuclear power plant. However, existing CFD subcooled boiling model was developed under high pressure condition [1]. In this study, it is confirmed that CFD can simulate low-pressure subcooled boiling using ANSYS CFX 2021 R1.

## 2. Methods and Results

The SUBO experiment is a representative low pressure subcooled boiling experiment, which was conducted by Korea Atomic Energy Research Institute (KAERI) [2]. In order to evaluate the accuracy of the existing RPI model at low pressure, the SUBO experiment is analyzed by CFX. Among many experiment cases, base case is modeled.

### *2.1 Modeling Method*

Fig. 1. shows the calculation domain and mesh in CFX and boundary conditions. For reducing computational resources, only a 5 degree angle was modeled using the symmetry condition in the flow channel cross section and only the heated section was simulated. The inner wall was considered as heat flux boundary, and the same heat flux as that of the test section was applied in experiment, and an adiabatic condition was assumed for the outer wall.

Eulerian model was used to model the two-phase flow, and water was assumed to be continuous fluid and vapor to be dispersed fluid. The wall boiling model was given by the RPI model. Additional models are summarized in Table I.

The convergence result is shown in Fig. 2. Residuals of momentum, mass, turbulence, and heat transfer converge well all smaller than $10^{-4}$ as the target. In addition, system imbalance converges well with a maximum error of 0.0022%.

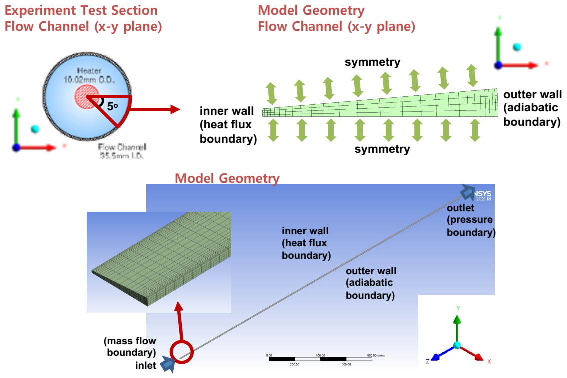

Fig. 1. Geometry and Mesh Information and Boundary Conditions.

Table I: Modeling Information

| Mesh information | |
|---|---|
| Number of nodes | 202,635 |
| Number of elements | 156,000 |
| Modeling information | |
| Fluid models | 2C, 6M, 2E, 1V |
| Turbulence | SST |
| Multiphase | Particle |
| Dispersed phase mean diameter | 2 mm |
| Interphase momentum | Ishii-Zuber |
| Interphase momentum | Lift Force (Saffman Mei)<br>Virtual Mass Force<br>Wall Lubrication Force (Antal)<br>Turbulent Dispersion Force (Favre Averaged) |
| Interphase mass | Thermal phase change |
| Interphase heat | Ranz-Marshall |
| Wall boiling | RPI model |
| Turbulence enhancement | Sato Enhanced Eddy Viscosity |
| Material Properties | IAPWS library |
| Analysis type | |
| Analysis type | Steady |
| Residual target | 1E-4 |
| Conservation target | 1E-4 |
| Solver information | |
| Advection scheme | High Resolution |
| Turbulence numerics | First order |

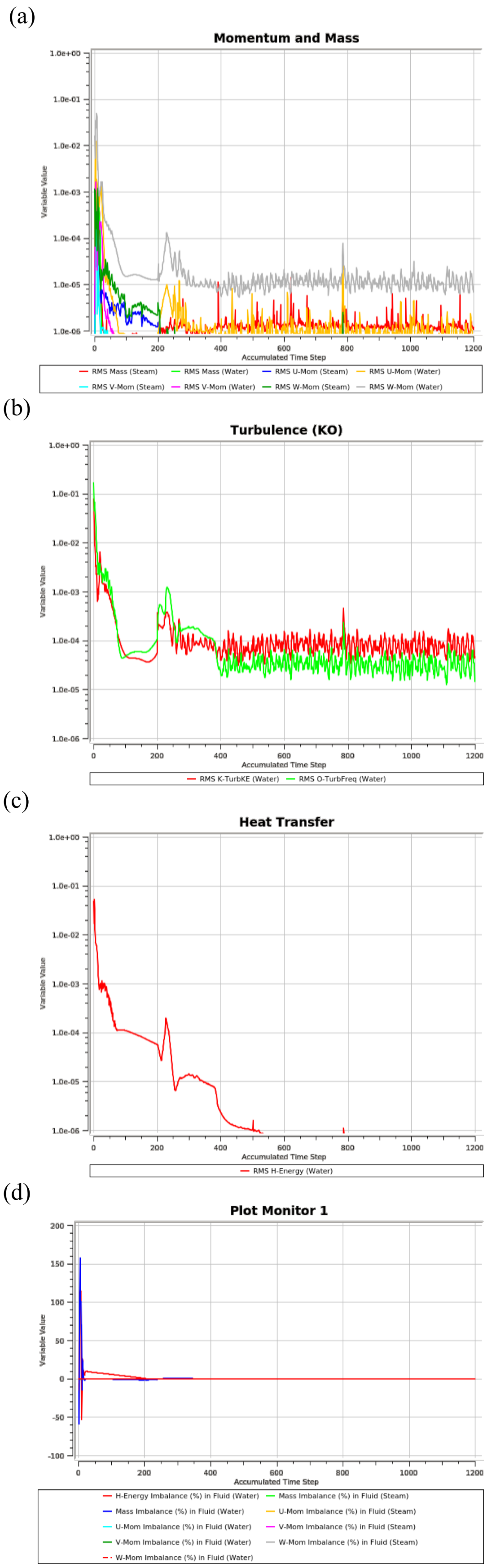

Fig. 2. Convergence Results (a) mass and momentum (b) turbulence (c) heat transfer (d) imbalance

## *2.2 Results*

Fig. 3. represents the void fraction in radial direction at each height. In the figure, dotted marks indicate experimental result, lines indicate simulation result, and the same color indicates the same height. It can be seen that the CFX analysis result predicts the void fraction lower than the SUBO experimental data. Although CFX predicts that the void fraction increases with height, the increase is small compared to the experiment. The void fraction near the heater rod was predicted lower than the experiment results at other positions except the lowest point. In addition, in this lowest point, there were no bubbles in the experiment, but it was predicted that the bubble was already generated in the CFX. Also, at the highest point in the heated surface, it has a peak near the heater rod in the experiment, but in the CFX analysis result, the void fraction has a peak at a position slightly shifted from the center.

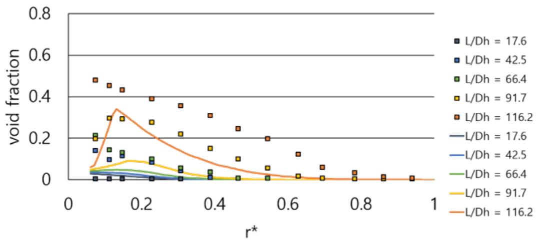

Fig. 3. Void Fraction in Experiment and CFD.

## 3. Conclusions

CFD approach to model complex two-phase flow will become very important for predicting severe accident consequences. However, existing CFD subcooled boiling model was developed under high pressure condition. In this study, it is confirmed that CFD can simulate low-pressure subcooled boiling but requires further improvement to have better accuracy. The SUBO experiment, which is a representative low pressure subcooled boiling experiment, is simulated with CFD. Experimental and CFD results under SUBO condition are quite different. It is necessary to improve the CFD boiling model, that is RPI model, under low pressure conditions.

## ACKNOWLEDGMENTS

This work was supported by the KAI-NEET Institute, KAIST, Korea. (N11220026). We would like to acknowledge the technical support from ANSYS Korea, South Korea.

## REFERENCES

[1] Choi, Y. S., et al. "CFD validation for subcooled boiling under low pressure," Journal of the Korean Society of Marine Engineering, 40(4), pp. 275-281 (2016).

[2] Yun, Byong-Jo, et al. "Experimental investigation of local two-phase flow parameters of a subcooled boiling flow in an annulus." Nuclear engineering and design 240.12 (2010): 3956-3966.