CFD Simulation of Pool Thermal Mixing

Y. T. Moon[a*], H. J. Ko[a], C. K. Park[b], C. H. Song[b], G. C. Park[c]
[a]*Korea Power Engineering Co., Inc, 360-9 Mabuk-Dong, Giheung-Gu, Yonin-Si, Gyeonggi-Do, 446-713, Korea.*
[b]*Korea Atomic Energy Research Institute (KAERI), Yuseong P.O. Box 105, Daejeon 305-600, Korea*
[c]*Seoul National University, San 56-1, Sillim-Dong, Kwanak-Gu, Seoul 151-742, Korea*
[*]*Corresponding author: moon@kopec.co.kr*

1. Introduction

According to the adoption of IRWST, one of the issues in questions is to ensure the efficient and stable condensation of steam discharge without resulting unstable steam bubble oscillation by knowing the local temperature in the IRWST.

Some studies have been carried out on thermal mixing phenomena in a subcooled water pool. For a thermal mixing CFD analysis, Kang, et al. used the calculated temperature and velocity of the condensed water and the entrained water from the condensation region model[1]. In the present paper, CFD simulation has been performed with the modified condensation region model. Further, these predictions have been compared with the cylindrical water pool test data performed by KAERI.

2. CFD calculation

The sophisticated computational analysis to understand the steam condensation process is desired but the state-of-the art computational techniques are still immature enough to provide reasonable results. The discharged steam into the subcooled water pool is condensed over a short distance so that we can look at the liquid exiting the condensation region for its effects on the pool thermal mixing. Therefore, we use the steam condensation region model in which the steam jet is condensed into water within the steam jet penetration length.

2.1. Cylindrical water pool experiment

The cylindrical water pool test was performed by modifying the Blowdown and Condensation (B&C) facility in KAERI. The sparger was made of a 1.25 in. schedule 40 pipe with 16 side discharge holes of 5 mm diameter, 8 LRR of 3.75 mm diameter, and a bottom hole of 4.7 mm diameter. To investigate the thermal mixing effects by the variation of steam discharge direction, three types of spargers were used in the tests. The steam is discharged through the side holes alone in type A sparger and through LRR alone in type B sparger. Finally, in type C sparger, all holes (side holes, LRR and bottom hole) are considered as a steam discharge area. The steam flux was varied from 300 to 900 kg/m^2-s, and the initial pool temperature was varied from 40 to 90 ^{o}C. The 166 thermocouples are arranged to measure the water temperature.

2.2. Condensation region model

The steam jet dimension plays a vital role in complex multiphase flows. The penetration length is defined as a function of the steam mass flux, a discharge hole geometry, and the temperature and pressure of water pool. Various authors proposed correlations describing the full condensation length (x) of steam jets[2,3]. These have the form

$$\frac{x}{d} = C_1 B^{C_2} \left(\frac{G}{G_{ref}} \right)^{C_3} \quad (1)$$

where the C's are coefficients that vary some from author to author. However, in view of the results obtained above for the jets created by the momentum of the injected steam, this condensation length does not seem to be of any particular importance. The steam will inject its momentum into the surrounding water no matter how short or long the condensation length. Gamble et al. in addressing a similar problem make mass and momentum balances in a cylindrical control volume[4]. For simplicity, we use a cylindrical control volume as a condensation region and assumed that the jet condensation length is 7 jet diameters. And the width of the jet can be calculated using Eq. (2) [1].

$$\frac{\text{Jet Width}}{x} = \tan 13^\circ \quad (2)$$

For each discharge hole, the condensation region model without considering the complicated velocity profiles is established as in Fig. 1. For the complicated geometry in a prototype multi-hole sparger, we use the lumped condensation region model, in which the discharge holes are put together in a single cylindrical shape. Fig. 2 shows the individual and lumped condensation region model.

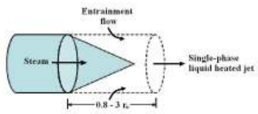
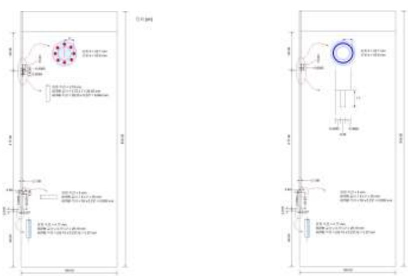


Fig. 1. Steam jet condensing at vent exit



(a) Individual (b) Lumped
Fig. 2. Steam Condensation Region Model

2.3. Mathematical models

Mass continuity is simply expressed by stating that the exit flow rate from condensation region is equal to the sum of the injected steam flow rate and the entrained flow rate

$$\dot{m}_s + \dot{m}_{entr} = \dot{m}_{cond} \qquad (3)$$

Momentum conservation can be written, considering the radial expansion of jet, as

$$\frac{G_s^2}{\rho_s} A_s = u_{cond}^2 \rho_{sat} A_{cond} \qquad (4)$$

where G is the mass flux. Defining the area ratio, $a_s = A_s / A_{cond}$, Eq. (4) can be solved for the velocity at the exit of the condensation region.

$$u_{cond}^2 = \frac{A_s}{A_{cond}} \frac{G_s^2}{\rho_s \rho_{sat}} \text{ and } u_{cond} = G_s \left[\frac{A_s}{A_{cond} \rho_s \rho_{sat}} \right]^{0.5} \qquad (5)$$

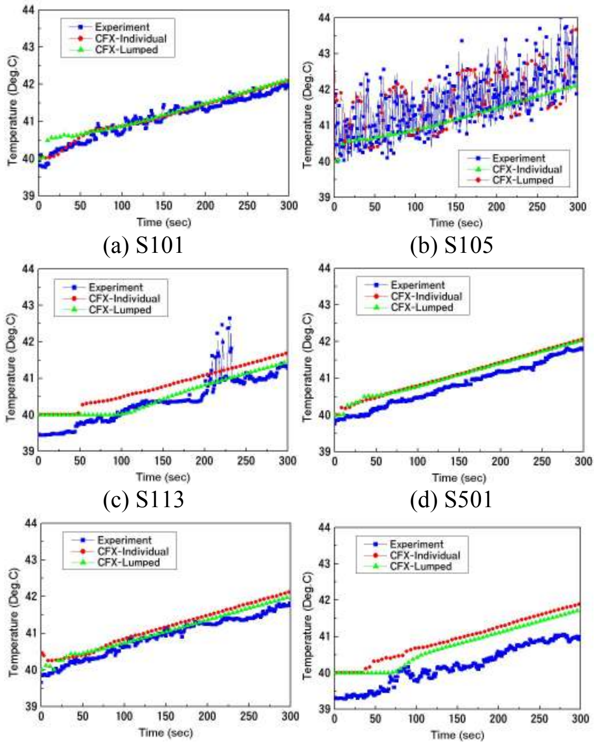
Knowing all the variables on the right side of this equation, we can compute the u_{cond}. In a individual condensation region model, the obtained velocity of condensed water and the mass and energy of the discharged steam are used as input of CFX calculations. In a lumped condensation region model, the geometry of the lumped region is determined based on the same condensed area of that in a individual condensation region model.

2.4. Grid model and boundary conditions

Since the flow pattern in the pool varies a little in the circumferential direction, the developed model is generated with an axisymmetric condition. A commercial software package CFX 11 has been used. For the individual condensation region model, a total of 420,000 hexagonal nodes have been used over the entire geometry. Transient simulations were carried out with a time step size of 0.1s with convergence criteria that the residual falls below 10^{-4}. Data points at all thermocouple locations have been tracked in order to verify the asymptotic state of the solution.

3. Results and discussion

This paper compared the results of the experiment with the CFX analysis in the type C sparger with 40℃ of initial pool temperature and 600 kg/m^2-s of steam mass flux. Fig. 3 shows the comparison of the temperature measurements with the CFX analysis results for 300s. The comparison of the calculated and experimentally measured temperature profiles shows similar trends as a whole. The comparison of temperature at the front of the jet flow (S105) shows that the experimental data are oscillating much little more than the calculated data and also the CFX could not simulate the temperature fluctuation phenomena. At other locations, the CFX predicts the observed trends of temperature variation well, but it overestimates the temperature at the upper part of the tank. In all cases, the trends of the temperature variation were very similar between the individual and the lumped condensation region models.



(a) S101 (b) S105
(c) S113 (d) S501
(e) S505 (f) S513

Fig. 3. Temperature profile in type C test

4. Conclusion

CFD simulations have been performed for understanding thermal mixing phenomena caused by the steam jet in the subcooled pool water. To decrease the computer memory by reducing the mesh number, the lumped condensation region model is introduced. A good agreement with the experimental data indicates the validity of the lumped steam condensation region model.

The success of the simulation demonstrated that CFD approaches are quite powerful to support the detailed physical understanding of the thermal mixing phenomena.

REFERENCES

[1] H. S. Kang and C. H. Song, "CFD Analysis for Thermal Mixing in a Subcooled Water Tank under a High Steam Mass Flux Discharge Condition", NED, Vol. 238, Issue 3, pp.492-501, 2008.
[2] Weimer, J.C., Faeth, G.M., Olson, D.R., 1973. Penetration of vapor jets submerged in subcooled liquids. Am. Inst. Chem. Eng. J. 19 (3), 552-558
[3] Kerney, P.J., Fathe, G.M., Olson, D.R., 1972. Penetration characteristics of submerged jet. American Institude of Chemical Engineering Journal 18 (3), 548-553
[4] Gamble, R.E., Nguyen, T.T., Shiralkar, B.S., Peterson, P.F., Greif, R. Tabata, 2001, H., Pressure suppression pool mixing in passive advanced BWR plants. NED-204, 321-336