Proceedings of the 10th World Congress on Momentum, Heat and Mass Transfer (MHMT 2025)
Barcelona, Spain - April 10 - 12, 2025
Paper No. ICMFHT 105
DOI: 10.11159/icmfht25.105

CFD Simulation of Loop Thermosiphon Using the Wall Boiling and Condensation Models

Quoc-Tuan Le [1], Byoung Jae Kim [1*]
[1]Department of Mechanical Engineering, Chungnam National University
99 Daehak-ro, Yuseong-gu, Daejeon, South Korea
bjkim@cnu.ac.kr

Extended Abstract

A two-phase loop thermosiphon is a passive heat transfer device whose working principle is based on natural circulation coupled with the phase change process of a fluid. In a large-scale heat exchanger where the environment is dominant by gravity, like a containment cooling system, this concept has been employed to facilitate the removal of core decay heat and maintain containment integrity during a Loss-of-Coolant scenario. 1-D system analysis codes have been widely used to predict thermal-hydraulic phenomena in the containment [1]. Although adequately accurate heat transfer can be estimated in a reasonable time, the above codes lack the ability to describe local flow effects. On the contrary, CFD codes, which can take account of two-phase flow behaviors, are powerful tools providing useful insight into such details. In the framework of the Euler-Euler two-fluid model (TFM), while the well-known wall heat flux partitioning model developed by Rensselaer Polytechnic Institute (RPI) for subcooled flow boiling has been extensively studied in the literature, only a limited number of wall condensation model for large-scale containment is presented. As a result, not much research has been carried out with the TFM approach.

A method proposed in [2] to incorporate the film condensation effect to a thermal hydraulic CFD code (CUPID) by introducing a sub-grid liquid film model to obtain wall and interfacial shear stresses along with interfacial area concentration and interfacial heat transfer coefficients, then transfer those values to two-fluid equations employed in CUPID, subsequently. In the present study, an incorporation between the conventional RPI wall boiling model for the heating section and wall film condensation based on heat and mass transfer analogy for the cooling part of a two-phase loop thermosiphon is proposed and implemented through User Define Function in a commercial CFD software (FLUENT). For simplicity, the noncondensable gas will be ignored, and species equations will not be solved. The boiling model developed was validated against the data of [3]. Meanwhile, the ongoing implementation of film condensation will be compared with the experimental data of [4] which only pure vapor cases are considered. Afterward, they will be incorporated to model a two-phase thermosiphon. This study is expected to contribute to the development of the multidimensional analysis of a two-phase thermosiphon applied in the design of Passive Containment Cooling System.

References

[1] J. Lee, Y. Park, S.-S. Jeon, J.-Y. Park, and H. K. Cho, "Assessment of MARS-KS prediction capability for natural circulation flow in passive heat removal system," *Nuclear Engineering and Technology*, Apr. 2024.

[2] J. Lee, G.-C. Park, and H. K. Cho, "Simulation of wall film condensation with non-condensable gases using wall function approach in component thermal hydraulic analysis code CUPID," *Journal of Mechanical Science and Technology*, vol. 32, no. 3, pp. 1015–1023, Mar. 2018.

[3] G.G. Bartolomej and V.M. Chanturiya, "Experimental Study of True Void Fraction When Boiling Subcooled Water in Vertical Tubes," *Thermal Engineering*, **14**, pp. 123–128 (1967) (translated from *Teploenergetika*, **14**(2), pp. 80–83 (1967)).

[4] W. Ambrosini, N. Forgione, F. Merli, F. Oriolo, S. Paci, I. Kljenak, P. Kostka, L. Vyskocil, J. R. Travis, J. Lehmkuhl, S. Kelm, Y.-S. Chin, and M. Bucci, "Lesson learned from the SARNET wall condensation benchmarks," Annals of Nuclear Energy, Vol. 74, pp. 153-164 (2014).