Transactions of the Korean Nuclear Society Spring Meeting
Jeju, Korea, May 7-8, 2026

Numerical Investigation of Thermal-Hydraulic Behavior in a Double-Tube Thermosyphon with Separated Vapor–Liquid Flow Paths

Hyunhwa Lee, Sewook Hwang, Kyung Mo Kim*
Department of Energy Engineering, Korea Institute of Energy Technology (KENTECH), Naju 58330, Republic of Korea
*Corresponding author: kmokim@kentech.ac.kr

***Keywords:** passive safety, thermosyphon, boiling/condensation, natural circulation, two-phase flow

1. Introduction

Existing decay heat removal systems in nuclear reactors generally consist of pumps, valves, and external power sources. However, under conditions such as station blackout or inappropriate operation of the valves, the decay heat removal through the safety systems, which depend on the external power sources or mechanical valves will be significantly limited. Therefore, another design candidate of the passive safety system, which is independent on the valves for the actuation and operation, can enhance the reactor safety. From this perspective, thermosyphon has been considered as an alternative device for the cooling of spent fuel pool, dry storage cask, containment, etc. [1, 2], because it actuates with the temperature difference and operates by natural circulation. However, in concentric thermosyphons, a counter-current flow occurs between the upward vapor flow and downward liquid film flow as they share flow path. The interfacial shear results in flooding and entrainment phenomena, which ultimately limit the maximum heat transport capacity [3]. As a result, the application of the thermosyphon in reactor scale is challenging.

To enhance the feasibility of thermosyphon-based passive safety systems by mitigating the flooding and entrainment phenomena, this study proposes a double-tube thermosyphon, which separates the flow paths of upward vapor flow and downward liquid flow. Due to the lack of experimental data demonstrating the flow separation and improved heat removal capacity of the double-tube thermosyphon, the thermal-hydraulic behavior inside the proposed thermosyphon design is preliminary analyzed with a volume-of-fluid (VOF)-based computational fluid dynamics (CFD) analysis is performed in this study. In addition, to identify the physical insights regarding the effects of design parameters on heat transfer capacity, a series of CFD analysis was carried out.

2. Design of double-tube thermosyphon

2.1 Working principle

Double-tube thermosyphon consists of an outer tube and an inner tube, with a tube separator installed. As illustrated in Fig 1, double-tube thermosyphon is divided into three primary sections: the evaporator at the bottom, the adiabatic section in the middle, and the condenser at the top.

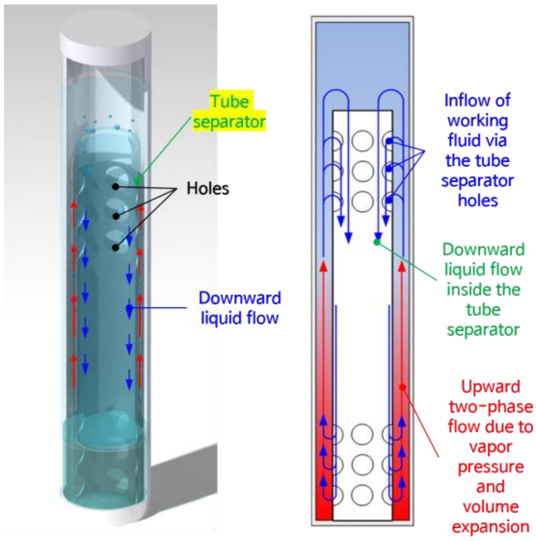

Fig 1. Double-tube thermosyphon structure

In the evaporator section, the working fluid absorbs heat and undergoes boiling, generating vapor. The produced vapor rises upward through the annular gap formed between the outer tube and the tube separator due to the hydrostatic pressure difference between the liquid and vapor phases. Upon reaching the condenser section, the vapor transfers heat to the cooling surface and condenses into liquid. The condensed liquid then passes through holes in the tube separator and enters the inner tube. Driven by gravity, the liquid flows downward through the inner tube and returns to the evaporator section, completing the circulation [4].

2.2 Design parameters

CFD simulations were conducted by systematically varying the principal design parameters to identify the optimal conditions for maximizing the heat removal capacity of a double-tube thermosyphon. Since the performance of a natural circulation thermosyphon is governed by the vapor generation rate, liquid return flow rate, and internal phase distribution, the design parameters directly influencing these mechanisms were selected for investigation.

The key design parameters examined in this study were the working fluid filling ratio (FR) and the internal pressure. At low FR, evaporator dry-out and intermittent liquid return occur due to insufficient liquid supply, whereas excessive FR reduces the effective vapor flow area and suppresses the overall circulation rate. Under low-pressure conditions, large vapor structures and intensified flow oscillations are observed, whereas elevated pressure stabilizes the vapor–liquid interface and promotes a stable natural circulation.

Based on a detailed analysis of the internal two-phase flow structures and circulation stability associated with variations in each design parameter, quantitative design guidelines were established to maximize the heat removal capacity.

3. Methods

3.1 Model and Conditions

To analyze the thermal-hydraulic phenomena inside the double-tube thermosyphon, a transient pressure-based solver was employed in conjunction with the VOF multiphase model. Phase change between liquid and vapor phases was modeled using the Lee evaporation–condensation formulation [5]. Turbulence effects were accounted for using the standard *k–ε* model, and gravitational acceleration was included to enable natural circulation driven by density differences. The thermophysical properties of water and vapor were obtained from the ANSYS Fluent material property database and treated as temperature-dependent.

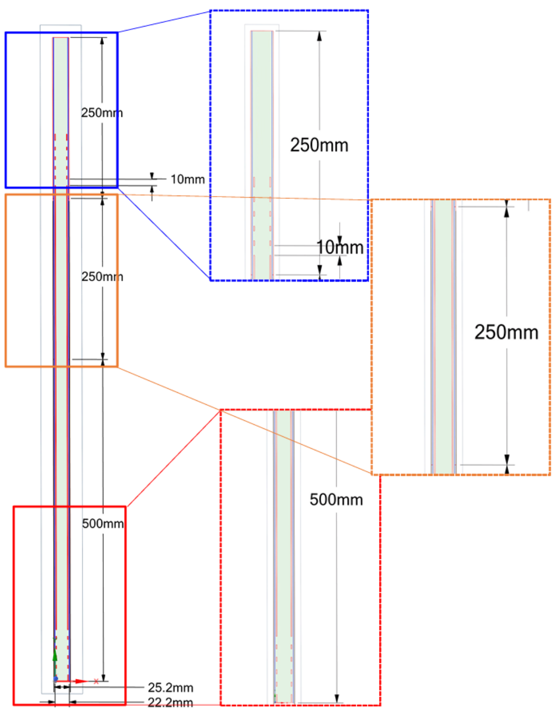

Fig 2. Fluent geometry model of the double-tube thermosyphon

The computational geometry of the double-tube thermosyphon is shown in Fig 2. The total length of the double-tube thermosyphon was 1 m, consisting of an evaporator section, an adiabatic section, and a condenser section with lengths of 500 mm, 250 mm, and 250 mm, respectively. A tube separator with a length of 700 mm was inserted inside the outer tube to structurally divide the flow passages. The separator was designed with an outer diameter of 19.05mm and an inner diameter of 17.05mm, forming an annular gap for vapor upflow and an inner tube for liquid return flow. Ten inlet holes with a diameter of 10 mm were drilled at both ends of the tube separator to allow the condensed liquid to enter the inner return channel at the top end and deliver the liquid to the annular gap at the bottom end. The reference simulation condition was defined with a working fluid FR of 50%, an internal pressure of 1 bar, and a heat flux of 57 kW/m².

Table 1. Mesh quality metrics of the double-tube thermosyphon CFD model

Category	Orthogonality	Skewness
Maximum	1	0.89237
Minimum	0.38045	1.3×10^{-10}
Average	0.97229	0.13028

The computational domain was discretized using structured finite-volume meshes. The grid size in the bulk flow region was approximately 0.3 mm, while a refined mesh with a minimum size of 0.1 mm was applied near the wall to accurately resolve phase change and thermal boundary layer effects. Inflation layers were introduced to adequately capture steep velocity and temperature gradients in the near-wall region. Mesh quality was quantitatively assessed using orthogonality and skewness metrics, as summarized in Table 1, confirming satisfactory numerical accuracy and grid reliability.

4. Results and discussion

4.1 Demonstration of flow separation

To confirm the formation of natural circulation, i.e., flow separation between the upward and downward flow, in accordance with the design concept of the proposed double-tube thermosyphon, the internal two-phase flow characteristics were investigated under the reference condition.

As shown in Fig 3, bubbles generated in the evaporator section ascended along the annular gap between the outer tube and the separator, the liquid in the condenser section entered the inner tube through the holes located at the upper end of the separator. Subsequently, a downward flow was established along the inner tube, forming a continuous liquid return path.

Fig 4 represents velocity fields and void fraction distributions at the evaporator, adiabatic, and condenser sections. The velocity vector distribution indicates that the time-averaged flow direction in the inner tube is downward. In addition, the upward flow velocity in the annular gap is higher than that in the inner tube due to the higher void fraction. The maximum velocity is observed in the adiabatic section, followed sequentially by the condenser and evaporator sections. This velocity distribution is consistent with the typical pressure profile of a thermosyphon, where the pressure is lowest in the adiabatic section and highest in the evaporator section, indicating that a stable natural circulation was established within the system. Therefore, it was confirmed that the double-tube configuration effectively separates the upward vapor flow and downward liquid flow paths, thereby reducing counter-current flow interaction and successfully achieve a stable natural circulation.

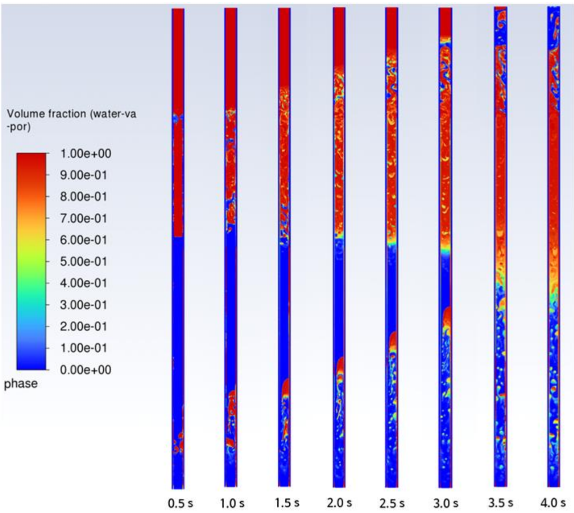

Fig 3. Transient thermo-hydraulic behavior inside the double-tube thermosyphon (internal pressure: 1 bar, filling ratio: 50%, heat flux: 57 kW/m²)

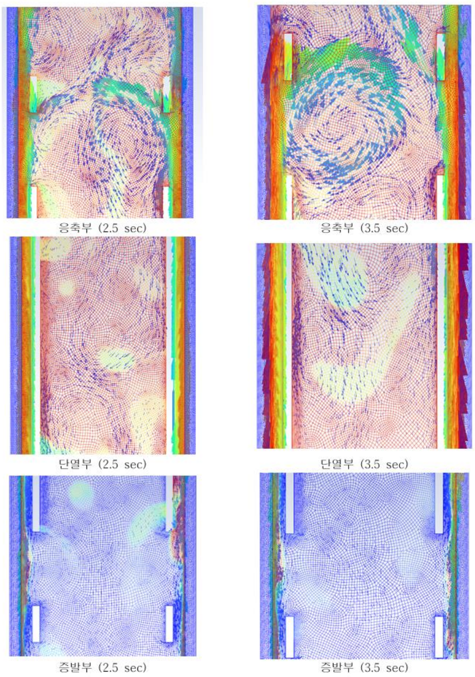

Fig 4. Localized liquid volume fraction and velocity distribution over time

4.2 Effect of Filling Ratio

Because one of primary design parameter determining the heat transfer performance of the thermosyphon is FR of the working fluid, variations of the hydraulic behavior with different FRs ranging from 30% to 70% are analyzed as shown in Fig 5 and 6.

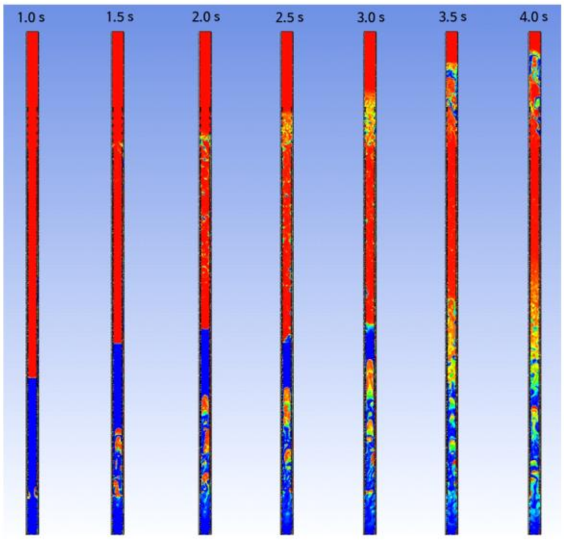

Fig 5. Temporal evolution of internal phase distribution in the double-tube thermosyphon at a working fluid filling ratio of 30% (heat flux: 57 kW/m², internal pressure: 1 bar)

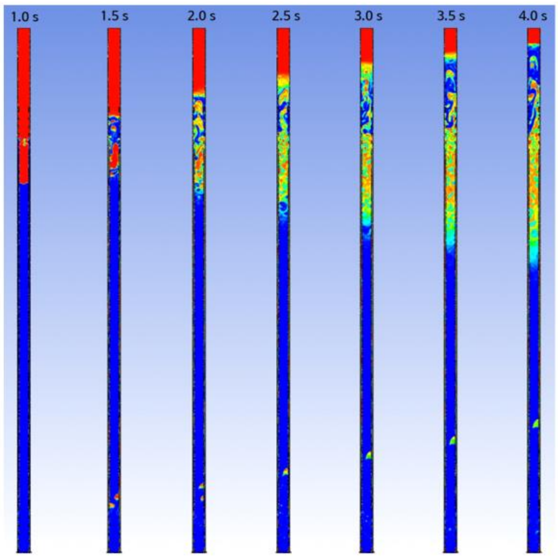

Fig 6. Temporal evolution of internal phase distribution in the double-tube thermosyphon at a working fluid filling ratio of 70% (heat flux: 57 kW/m², internal pressure: 1 bar)

Fig 5 illustrates the flow behavior under the 30% FR condition. In this case, insufficient liquid inventory in the evaporator section led to the rapid formation of a vapor core from the early stage of operation, and the downward liquid film was not continuously maintained. The liquid fragmented into droplets, resulting in intermittent return flow and unstable boiling behavior.

The flow characteristics at a FR of 70% are provided in Fig 6. From the initial stage, a continuous and thick liquid pool was formed in the evaporator section and the lower region of the adiabatic section. Bubble generation was primarily confined to the annular gap between the outer tube and the separator. Due to the increased liquid inventory, the hydrostatic pressure within the liquid pool increased, resulting in an elevated saturation temperature. Consequently, smaller bubble sizes and suppressed bubble coalescence were observed. In addition, as the separator region became predominantly filled with liquid, the effective vapor rising pathway was relatively reduced, leading to localized concentration of vapor flow. As a result, the driving force for natural circulation decreased, indicating the potential reduction of circulation flow rate. Therefore, although the high FR condition enhances operational stability, it simultaneously increases vapor discharge resistance and reduces circulation flow rate, suggesting a possible deterioration in overall heat transfer performance.

4.3 Effect of Internal Pressure

As the internal pressure affects the thermophysical properties of working fluid and two-phase flow characteristics, it .

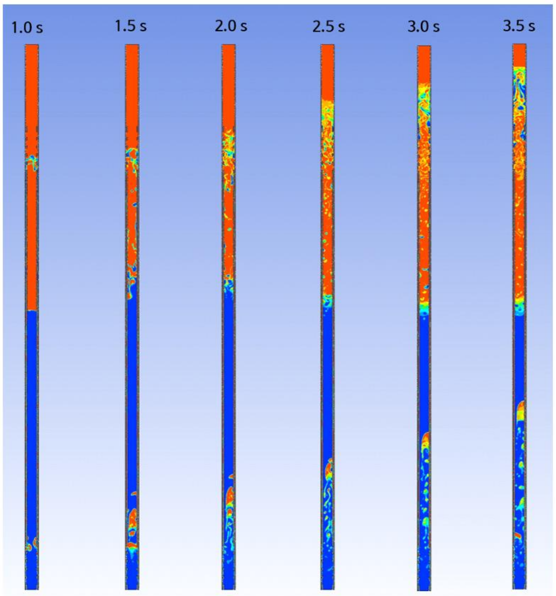

Fig 7. Temporal evolution of internal phase distribution in the double-tube thermosyphon at an internal pressure of 0.5 bar (heat flux: 57 kW/m², filling ratio: 50%)

The flow behavior under the 0.5 bar condition is detailed in Fig 7. At the pressure below the atmospheric pressure, the latent heat increases and vapor density decreases, resulting in an increased wall superheat and the formation of considerably larger bubbles. The larger bubble growth and pronounced interfacial motion led to significant pressure fluctuations, indicating a tendency toward reduced flow stability.

A contrasting behavior at 3 bar is documented in Fig 8. With the increase in saturation temperature and the reduction in vapor–liquid density difference, smaller and more uniformly distributed bubbles were formed under the equivalent heat load. A relatively smooth interface characteristic of annular flow developed. The lower liquid pool maintained its continuity over an extended period, and the amplitude of interfacial oscillation was markedly reduced compared to the low-pressure condition.

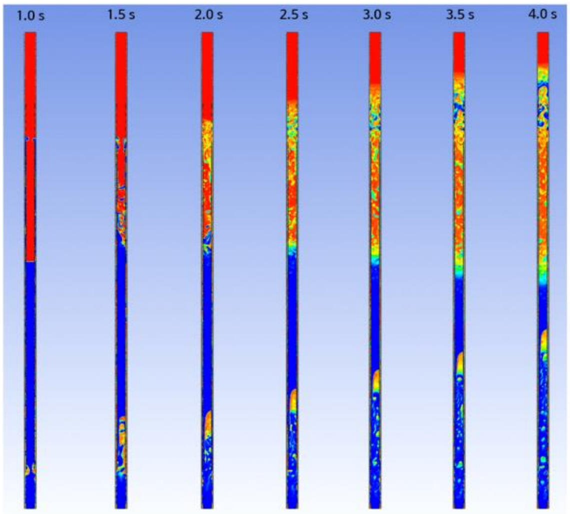

Fig 8. Temporal evolution of internal phase distribution in the double-tube thermosyphon at an internal pressure of 3 bar (heat flux: 57 kW/m², filling ratio: 50%)

5. Conclusions

In this study, the internal thermal-hydraulic behavior of a double-tube thermosyphon incorporating vapor–liquid flow separation was proposed to improve the applicability of thermosyphon toward the passive safety system by mitigating the limited heat transfer capacity of the concentric designs. The CFD analysis results successfully demonstrated that vapor ascends through the annular gap while liquid containing entrained bubbles descends through the inner tube, thereby mitigating the interfacial shear between the vapor and liquid flows. Comparative analysis of FR and internal pressure provided an insight for the effects of design parameters on hydraulic behaviors inside the device.

In the future, further investigation will be carried out assessing the long-term operational stability, heat transfer performance, and maximum heat transfer capability under extended operating conditions and additional design parameters, to figure out the design guideline achieving the optimal heat transfer performance.

ACKNOWLEDGEMENT

This work was supported by the National Research Foundation of Korea (NRF) grant funded by the Korea government (MSIT) (No. RS-2025-02654004).

REFERENCES

[1] Castro, S.I.C., Kirsch, M., Kulenovic, R. et al. Experimental investigation of the heat transfer characteristics, operating limits, and temperature distribution of a prototypically 3 m long two-phase closed thermosyphon for spent fuel pool passive cooling. Exp. Comput. Multiph. Flow 6, 229–241 (2024). https://doi.org/10.1007/s42757-024-0193-2

[2] Y. S. Jeong and I. C. Bang, "Hybrid heat pipe based passive cooling device for spent nuclear fuel dry storage cask," Applied Thermal Engineering, vol. 96, pp. 277–285, 2016. https://doi.org/10.1016/j.applthermaleng.2015.11.086

[3] Kim, K. M., & Bang, I. C. (2016). Comparison of flooding limit and thermal performance of annular and concentric thermosyphons at different fill ratios. Applied Thermal Engineering, 99(Complete), p. 179–188. https://doi.org/10.1016/j.applthermaleng.2015.12.137

[4] Imura, H., Yoshida, M., "Heat transfer characteristics in two-phase double-tube thermosyphons," Transactions of the JSME, Series B, Vol. 56, No. 532, 1990.

[5] Lee, W. H., "A Pressure Iteration Scheme for Two-Phase Flow Modeling," Technical Report LA-UR 79-575, Los Alamos Scientific Laboratory, 1979.