

Article

CFD Implementation and Preliminary Validation of a Combined Boiling Model (CBM) for Two-Phase Closed Thermosyphons

Jure Štrucl ¹ , Jure Marn ² and Matej Zadavec ^{2,*} 

¹ AVL-AST d.o.o., Ulica Kneza Koclja 22, SI-2000 Maribor, Slovenia; jure.strucl@gmail.com

² Faculty of Mechanical Engineering, University of Maribor, Smetanova Ulica 17, SI-2000 Maribor, Slovenia

* Correspondence: matej.zadavec@um.si

Abstract

Predicting phase-change heat transfer in two-phase closed thermosyphons (TPCTs) represents a significant challenge owing to the complex interaction of boiling, condensation, and conjugate heat transfer (CHT) mechanisms. This study presents a numerical investigation of a TPCT using the Combined Boiling Model (CBM) within a conjugate heat transfer (CHT) framework. Unlike prior TPCT studies, the CBM integrates an improved RPI-based wall boiling model with sliding bubble dynamics, a laminar film condensation closure, and Lee-type bulk phase change in a single, energy-consistent formulation suited for engineering-scale meshes and time-steps. Building on these extensions, we demonstrate the approach on a vertical TPCT with full CHT and validate it against experiments and a VOF–Lee reference. Simulations for heat loads ranging from 173 to 376 W capture key flow features, including vapour generation, vapour-pocket dynamics, and thin-film condensation, while reducing temperature deviations typically below 3% in the evaporator and adiabatic sections and about 2 to 5% in the condenser. The results confirm that the CBM provides a physically consistent and computationally efficient approach for predicting evaporation–condensation phenomena in TPCTs.

Keywords: heat pipe; thermosyphon; multiphase; conjugate heat transfer; Combined Boiling Model (CBM); Euler–Euler



Academic Editor: Kambiz Vafai

Received: 22 October 2025

Revised: 8 November 2025

Accepted: 11 November 2025

Published: 13 November 2025

Citation: Štrucl, J.; Marn, J.; Zadavec, M. CFD Implementation and Preliminary Validation of a Combined Boiling Model (CBM) for Two-Phase Closed Thermosyphons. *Fluids* **2025**, *10*, 296. <https://doi.org/10.3390/fluids10110296>

Copyright: © 2025 by the authors. Licensee MDPI, Basel, Switzerland. This article is an open access article distributed under the terms and conditions of the Creative Commons Attribution (CC BY) license (<https://creativecommons.org/licenses/by/4.0/>).

1. Introduction

Modern devices such as electric motors, hydrogen fuel cells, control units, motherboards, and batteries [1,2], particularly in the automotive and aerospace industries, are becoming increasingly compact and powerful. Consequently, they generate more heat and therefore require adequate cooling to ensure reliable operation. As the demand for compact and powerful electronic systems rises, simple single-phase cooling systems are no longer sufficient for their thermal management; instead, multiphase cooling systems that incorporate phase-change processes are required.

Such cooling systems include, among others, heat pipes, which have been widely used and studied because of their ability to transfer large amounts of heat while maintaining minimal temperature gradients. First investigated in the 19th century [1] and formalised by Grover in 1963 [2], heat pipes exploit evaporation and condensation to achieve exceptionally high effective thermal conductivity. Predicting their thermal-fluid behaviour, however, is challenging due to the coupled effects of multiphase flow, compressibility, and conjugate heat transfer. Early analytical models treated heat pipes as one-dimensional conduction systems with simplified assumptions for fluid transport [3], but modern studies increasingly

employ computational fluid dynamics (CFD) to resolve vapour flow, liquid film dynamics, and interfacial phase change [4].

In its simplest form, a heat pipe is a TPCT as shown in Figure 1.

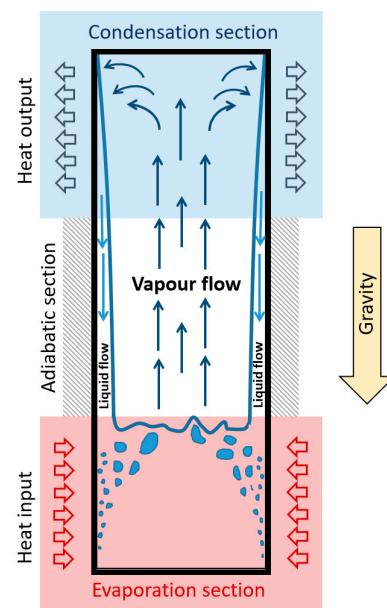


Figure 1. Schematic of the complete thermosiphon cycle.

Despite the apparent simplicity, a TPCT is numerically very complex to model, as accurate simulations must account for nucleate boiling, condensation, and evaporation processes. Like many high-performance cooling devices, TPCTs achieve their heat transfer capability by operating in the nucleate-boiling regime; however, this regime exists only over a narrow wall superheat band (e.g., $\Delta T \approx 1\text{--}30\text{ }^{\circ}\text{C}$ for water at atmospheric pressure), imposing strict modelling requirements [5,6].

Various CFD studies on subcooled wall boiling have been reported in the literature [7–9]. These reviews indicate that researchers primarily employ interfacial surface tracking methods, such as Volume-of-Fluid (VOF), or the Euler–Euler framework, to model boiling phenomena. VOF enables detailed bubble- and droplet-scale studies but is computationally expensive [10–13], whereas Euler–Euler, based on averaged field equations, allows for coarser grids and larger time-steps, making it computationally more efficient and thus more suitable for practical engineering applications.

In the Euler–Euler framework, researchers commonly model wall boiling using the RPI wall boiling model [14]. However, this model often requires empirical recalibration and may under-predict flow boiling conditions [15–18]. Further improvements are therefore needed, particularly for TPCT modelling, where circulation is driven by a combination of pressure and gravity effects. In this work, flow boiling RPI enhancements reported by Gilman [19] and Shi [20] were incorporated into the CBM model to extend its applicability to forced convective flow.

Furthermore, the wall boiling modelling in TPCT, bulk flow evaporation and condensation also play an important role. In previous studies, authors often employed the Lee model [11,15–17,21] to simulate TPCT, which has shown good agreement with experiments [6]. For this reason, the Lee model was also adopted as a component of the CBM.

When predicting the cooling performance of a system, such as one cooled by a TPCT, we are typically interested in the temperature within the solid component being cooled, that is, the wall of the device, and not only in the temperature at the inner wall of the TPCT. To accurately predict this temperature in the cooled material, conjugate heat transfer (CHT)

between the fluid and solid domains must be included; however, only a few studies have treated multiphase CHT comprehensively [9,18].

In previous work, we proposed a new state-of-the-art Combined Boiling Model (CBM), which integrates wall boiling based on an improved RPI formulation for forced convective flow, film condensation, and bulk phase change within an Euler–Euler multiphase framework. The model was first validated on a simplified CHT test case [22], showing good agreement between experiment and simulation. In the present study, we further improve the model by refining the bubble sliding treatment and by adding a wall condensation model, both of which are required to predict the thermal behaviour of a TPCT. According to our review of previous research, a model combining these sub-models within an Euler–Euler framework for TPCT thermal analysis has not yet been reported.

Why a Combined Boiling Model (CBM)?

To treat a TPCT as a true conjugate heat transfer problem in an Euler–Euler framework, all governing processes must be represented. At the walls, the modified RPI model partitions the wall heat flux and generates evaporative mass in the evaporator, while a Nusselt-type laminar film condensation model describes heat removal in the condenser; together, they provide a consistent description of mass and heat exchange at the wall. Once the phases detach and travel through the core, evaporation and condensation must also be modelled in the bulk at a finite rate; the Lee model supplies this by predicting vapour growth and collapse under local superheat or subcooling, which is essential for correct transport from the evaporator to the condenser. Because device performance is determined by solid temperatures and extracted heat, a two-sided CHT treatment is used as follows: sensible heat is transferred via a Robin-type interfacial condition with an interfacial heat transfer coefficient, and latent heat is applied on the solid side while the corresponding saturated mass is injected into the fluid. This energy-consistent placement avoids latent heat double counting. Consequently, combining RPI, film condensation, and Lee within one CBM is necessary for reliable prediction of TPCT behaviour at engineering mesh sizes and time-steps.

2. Model Description

2.1. Background

The CBM model was developed and preliminary validation on a simplified case was employed in our previous study [22]. That case represented an open system with a velocity inlet and a pressure outlet boundary condition, consisting of two domains: the fluid region and a solid heating plate. A detailed grid independence and time-step sensitivity analysis was performed, from which the optimal numerical parameters were established. These parameters were subsequently applied to the more complex heat pipe model investigated in the present work.

In contrast to the Robinson-type duct of the previous study, the present model represents a closed vertical heat pipe filled with working fluid. It comprises three main components: the fluid domain of the pipe, the solid copper casing, and the coolant region at the condenser side, see Figure 2.

Geometrically, this configuration is not significantly more complex than the Robinson test duct; however, from a numerical perspective, it is considerably more challenging. In particular, the closed domain requires consideration of vapour compressibility. Furthermore, bubble dynamics are no longer dictated by inlet velocity, but instead result from buoyancy forces, pressure gradients, and bubble drag within the flow, all of which must be explicitly resolved.

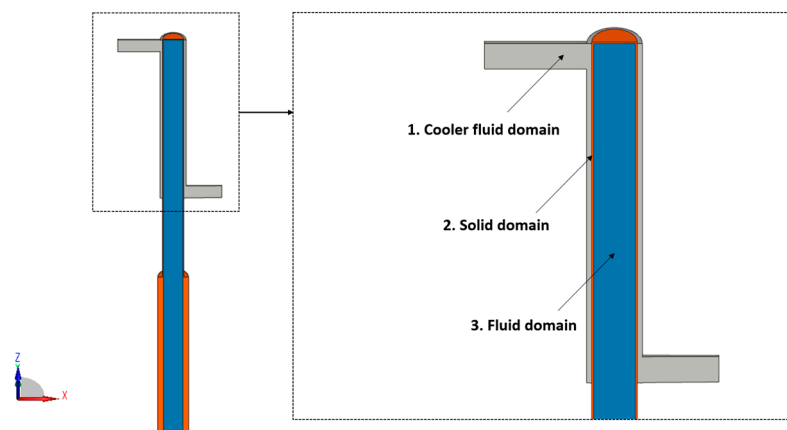


Figure 2. Model components (computational domains).

In addition, heat pipe simulations must account for condensation processes. For this reason, a simplified film condensation model has been incorporated into the CBM. The key extensions relative to the original analysis are summarised as follows:

- Closed system,
- Weakly compressible vapour flow,
- Inclusion of condensation,
- Three interacting domains: the solid domain (single-phase, energy equation), the heat pipe fluid domain (multiphase), and the coolant domain (single-phase flow).

2.2. Modelling Approach: Euler–Euler

For CFD simulations of boiling, especially subcooled flow boiling and condensation, two approaches are used most frequently as follows: interface-resolving methods, most notably the VOF approach [10–13], and averaged multiphase methods, such as the Euler–Euler two-fluid model [15–18]. VOF tracks or reconstructs a sharp liquid–vapour interface and is therefore well suited for studies of individual bubble growth, coalescence, and capillary effects, but it requires very fine grids and small time-steps to maintain interface quality, particularly in three-dimensional domains and for long transients. Consequently, applying VOF directly to cases with CHT, multiple interacting domains, and long operating times is computationally costly.

In the present work, an Euler–Euler two-fluid (gas–liquid) formulation is adopted as the base framework. In this approach, both phases are treated as interpenetrating continua that share the same pressure field, while phase interaction is represented through closure relations for interfacial momentum, heat, and mass transfer. Local phase change is not obtained from an explicitly captured interface, but from mechanistic sub-models. This makes the method more robust for flows with frequent phase appearance or disappearance, dispersed bubbles, and simultaneous wall boiling and condensation. For this reason, it has already been successfully applied to thermal problems such as quenching, spray quenching, boiling, and flow boiling [23–25], and it was also used in our previous work, where the CBM was first introduced [22]. Figure 3 shows the main differences between the Euler–Euler and VOF approaches. In the Euler–Euler approach, the vapour bubbles are represented as a continuous vapour volume fraction field whose behaviour depends on the chosen interfacial drag models, whereas the VOF approach explicitly tracks the liquid–vapour interface (shown green in the figure) and enforces the jump in properties from one phase to the other across that interface.

The proposed CBM is specifically constructed to compensate for the lack of an explicitly resolved interface in Euler–Euler simulations. It augments the two-fluid equations with wall boiling, film condensation, and bulk Lee-type source terms, so that

the correct heat-partitioning and phase-generation mechanisms are still reproduced at engineering-type mesh resolutions (≈ 1 mm) and practical time-steps ($\Delta t = 0.005$ s). Compared with VOF–Lee formulations reported in the literature, the present approach offers a substantially lower computational cost while retaining the capability to predict vapour-pocket formation, liquid film drainage, and condenser wall cooling, which are the key phenomena of interest in TPCT analysis.

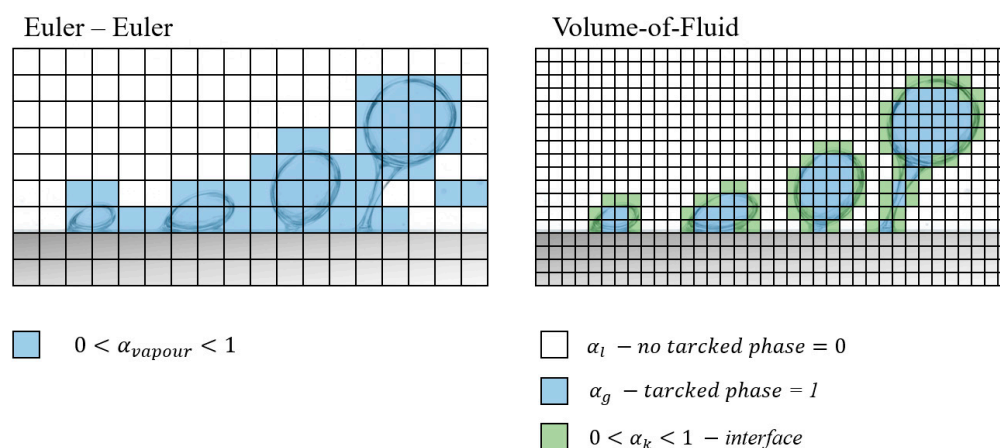


Figure 3. Comparison of Euler–Euler and VOF modelling.

In the Euler–Euler framework, the interface is represented in a volumetric sense by an interfacial area density, not as a sharp surface. Consequently, liquid and vapour can have different bulk temperatures, with sensible heat exchanged through an interfacial heat transfer term. Local thermal equilibrium is applied only within an effective interfacial layer for the evaluation of phase-change sources (RPI at the wall and Lee in the bulk), while the bulk remains thermally non-equilibrium.

2.3. Governing Equations

The Euler–Euler multi-fluid framework treats each phase as an interpenetrating continuum, with interphase exchange terms added to the conservation laws [26]. Ensemble averaging removes microscopic interfaces and yields macroscopic equations analogous to single-phase flow, extended by phase volume fractions and interfacial transfer terms [27–29]. Based on the formulation of Drew and Passman [30], the governing equations are

Continuity equation (phase k):

$$\frac{\partial \alpha_k \rho_k}{\partial t} + \nabla \cdot \alpha_k \rho_k \mathbf{v}_k = \sum_{l=1, l \neq k}^N \Gamma_{kl}, \quad k = 1, \dots, N \quad (1)$$

Here, α_k is the volume fraction of phase k , ρ_k the density, and v_k the velocity. The volume fraction compatibility condition is

$$\sum_{k=1}^N \alpha_k = 1 \quad (2)$$

Momentum equation (phase k):

$$\frac{\partial \alpha_k \rho_k \mathbf{v}_k}{\partial t} + \nabla \cdot \alpha_k \rho_k \mathbf{v}_k \otimes \mathbf{v}_k = -\alpha_k \nabla p + \nabla \cdot \alpha_k (\boldsymbol{\tau}_k + \boldsymbol{\tau}_k^t) + \alpha_k \rho_k \mathbf{f} + \sum_{l=1, l \neq k}^N \mathbf{M}_{kl} + \sum_{l=1, l \neq k}^N \mathbf{v}_{kl} \Gamma_{kl}, \quad k = 1, \dots, N \quad (3)$$

Here, τ_k and τ_k^t denote the viscous and turbulent stress tensors; M_{kl} is the interfacial momentum exchange (drag, lift, wall lubrication, and turbulent dispersion; Section 2.5); and $\mathbf{f}$ includes gravity and inertial forces. Pressure is assumed equal for all phases:

$$p = p_k, \quad k = 1, \dots, N \quad (4)$$

Energy equation (mixture form):

$$\begin{aligned} \frac{\partial \alpha_k \rho_k h_k}{\partial t} + \nabla \cdot \alpha_k \rho_k \mathbf{v}_k h_k = & \nabla \cdot \alpha_k (\mathbf{q}_k + \mathbf{q}_k^t) + \alpha_k \rho_k q_k''' + \alpha_k \rho_k \mathbf{f} \cdot \mathbf{v}_k + \\ & \nabla \cdot \alpha_k (\tau_k + \tau_k^t) \cdot \mathbf{v}_k + \alpha_k \frac{\partial p}{\partial t} + \sum_{l=1, l \neq k}^N H_{kl} + \sum_{l=1, l \neq k}^N h_k \Gamma_{kl} \end{aligned} \quad (5)$$

Here, h_k is the specific enthalpy of phase k ; $\mathbf{q}_k$ and $\mathbf{q}_k^t$ are the conductive and turbulent heat fluxes; H_{kl} is the sensible interfacial heat exchange between phases k and ℓ ; and Γ_{kl} is the mass transfer rate from k to ℓ . The term $h_k \Gamma_{kl}$ represents the latent heat source or sink.

Equation (2) enforces the volume fraction compatibility. Sensible interfacial heat exchange H_{kl} is closed in Section 2.4; wall boiling and condensation at boundaries are described in Sections 2.6 and 2.7; and bulk phase change is given in Section 2.8.

2.4. Energy

In addition to the mass and momentum conservation equations, the mixture energy equation (Equation (5)) is solved for the total (sensible + latent) enthalpy of the two-fluid system. Thermal non-equilibrium is allowed in the bulk; the liquid and vapour phases may have different temperatures and exchange sensible heat via an interfacial heat transfer term. Local thermal equilibrium (LTE) is enforced only within the effective interfacial layers used to evaluate phase-change sources, i.e., the modified RPI wall boiling layer at heated walls and the Lee-type bulk phase-change closure (where temperatures are locally relaxed to T_{sat} for source evaluation) [28]. This preserves correct heat partitioning for phase change while permitting bulk temperature differences to relax through interfacial convection–conduction.

Interfacial sensible heat exchange is modelled according to the Ranz–Marshall correlation [31]:

$$H_{cd} = h_i a_i (T_d - T_c) \quad (6)$$

where a_i is the interfacial area density and h_i the interfacial heat transfer coefficient. T_c and T_d are the bulk temperatures of the continuous and dispersed phases, respectively. For dispersed bubbly/droplet regimes, we use the following:

$$a_i = \frac{6\alpha_d}{d_{32}} \quad (7)$$

Here, d_{32} is the Sauter mean diameter, and α_d the dispersed-phase volume fraction.

$$h_i = \frac{\text{Nu} k_c}{d_{32}} \quad (8)$$

In the interfacial heat transfer coefficient, k_c is the thermal conductivity of the continuous phase and Nu is the Nusselt number, written as

$$\text{Nu} = 2 + 0.6 \text{Re}_d^{1/2} \text{Pr}_c^{1/3} \quad (9)$$

Here, Re_d and Pr_c are the Reynolds and Prandtl numbers of the continuous phase, defined as

$$\text{Re}_d = \frac{\rho_c |\mathbf{v}_d - \mathbf{v}_c| d_{32}}{\mu_c} \quad (10)$$

$$\text{Pr}_c = \frac{\mu_c c_{p,c}}{k_c} \quad (11)$$

Here, ρ_c is the continuous-phase density, $\mathbf{v}_d$ and $\mathbf{v}_c$ are phase velocities; $|\mathbf{v}_d - \mathbf{v}_c|$ is the slip velocity magnitude; μ_c is the continuous-phase viscosity; and $c_{p,c}$ is its specific heat.

Further details of the multi-fluid averaging framework are given in [26,29]. Details about latent enthalpy models, namely the modified RPI wall boiling, the Lee bulk phase change, and wall condensation, are explained in following Sections 2.6–2.8.

2.5. Interfacial Momentum Exchange

The interfacial momentum source term accounts for drag, turbulent dispersion, lift, and wall lubrication:

$$M_{kl} = F_D + F_{TD} + F_L + F_{WL} \quad (12)$$

The drag contribution is given by

$$F_D = \frac{3}{4} C_D \rho_c \alpha_d a_i |\mathbf{v}_r| \mathbf{v}_r \quad (13)$$

where $\mathbf{v}_r = \mathbf{v}_d - \mathbf{v}_c$ is the slip (relative) velocity between the dispersed-phase d and the continuous-phase c ; and C_D is the drag coefficient. The interfacial area density is denoted a_i (see Equation (7)).

In the present work, the regime-dependent drag C_D is calculated using the momentum interfacial exchange “Gas–Liquid System 3” [26,32]. It blends bubbly and droplet drag correlations based on dispersed-phase volume fraction, using Schiller–Naumann [33] or Tomiyama [34] models depending on bubble size and Eötvös number.

The Schiller–Naumann [33] drag model is defined as

$$C_D = \begin{cases} \frac{24}{\text{Re}_d} \left(1 + 0.15 \text{Re}_d^{0.687} \right) & \text{Re}_d \leq 1000 \\ 0.438 & \text{Re}_d > 1000 \end{cases} \quad (14)$$

For bubble flows where the bubble diameter is greater than 1 mm, the Tomiyama [34] drag model is more suitable, as follows:

$$C_D = \max \left\{ \min \left[\frac{24}{\text{Re}_d} \left(1 + 0.15 \text{Re}_d^{0.687} \right), \frac{72}{\text{Re}_d} \right], \frac{8}{3} \frac{Eo}{Eo + 4} \right\} \quad (15)$$

The Bubble Eötvös number (Eo) is defined as

$$Eo = \frac{g(\rho_l - \rho_g) D_d^2}{\sigma} \quad (16)$$

where $(\rho_l - \rho_g)$ is the difference between the liquid and gas densities, g is the acceleration due to gravity, D_d is the dispersed-phase (bubble) diameter, and σ is the liquid surface tension.

Turbulent dispersion accounts for vapour diffusion in mixing flows.

The “Gas–Liquid System 3” interfacial momentum exchange model defines the drag force between the gas and liquid phases as a function of the local volume fraction of the dispersed phase. Following Mimouni [32], the drag force is evaluated according to the prevailing flow regime:

- If the gas volume fraction α_2 is below the critical value for bubbly flow α_{bub} , the bubbly drag coefficient C_D^{bub} is used;
- If α_2 is above the critical value for droplet flow α_{drpl} , the droplet drag coefficient C_D^{drpl} is used;

- For values of α_2 between these two critical volume fractions, the drag force is a mixture of the bubbly and droplet drag forces.

Here, α_{bub} and α_{drpl} denote the model's bubbly and droplet critical dispersed-phase volume fraction thresholds used to switch/blend the drag closures. F_D is a mixture of the bubbly and droplet drag forces:

$$F_D^{(1,2)} = \begin{cases} F_D^{bub} & \alpha_2 < \alpha_{bub} \\ F_D^{mix} & \alpha_{bub} \leq \alpha_2 \leq \alpha_{drpl} \\ F_D^{drpl} & \alpha_2 > \alpha_{drpl} \end{cases} \quad (17)$$

where d_{bub} and d_{drpl} are the bubble and droplet diameters and the drag forces are given as

$$F_D^{bub} = \alpha_2 \rho_1 \frac{3 C_D^{bub}}{4 d_{bub}} |\mathbf{v}_2 - \mathbf{v}_1| (\mathbf{v}_2 - \mathbf{v}_1) \quad (18)$$

$$F_D^{drpl} = \alpha_1 \rho_2 \frac{3 C_D^{drpl}}{4 d_{drpl}} |\mathbf{v}_2 - \mathbf{v}_1| (\mathbf{v}_2 - \mathbf{v}_1) \quad (19)$$

$$F_D^{mix} = \frac{\alpha_{drpl} - \alpha_2}{\alpha_{drpl} - \alpha_{bub}} F_D^{bub} + \frac{\alpha_2 - \alpha_{bub}}{\alpha_{drpl} - \alpha_{bub}} F_D^{drpl} \quad (20)$$

2.6. Wall Boiling Model

Near-wall heat transfer is modelled using the RPI framework of Kurul and Podowski [14], extended for forced convective boiling following Gilman [19] with simplifications by Shi and Zhang [20]. The complete formulation was presented in our previous work [22]; therefore, only the governing partitioning and key correlations will be presented in this section.

The total wall heat flux is

$$\dot{q}_w = \dot{q}_{conv} + \dot{q}_{sta} + \dot{q}_{sli} + \dot{q}_{eva} \quad (21)$$

The evaporative component is

$$\dot{q}_{eva} = \dot{m} h_{fg}, \quad \dot{m} = \frac{\pi}{6} D_d^3 N_w \rho_v f \quad (22)$$

with bubble departure frequency f [35], nucleation site density N_w [36], and departure diameter D_d [37] from standard correlations.

The convective contribution is

$$\dot{q}_{conv} = C_m h_{conv} (T_w - T_l) (1 - A_b) \quad (23)$$

where C_m is the convective partition factor, h_{conv} the single-phase wall heat transfer coefficient, and $A_b = A_{sta} + A_{sli}$ the boiling area fraction.

Stationary and sliding bubble quenching fluxes are

$$\dot{q}_{sta} = h_{quen} (T_w - T_l) A_{sta}, \quad \dot{q}_{sli} = h_{quen} (T_w - T_l) A_{sli} \quad (24)$$

where h_{quen} is the transient conduction coefficient [38].

An important aspect of these correlations is the determination of the areas of influence, A_{sta} and the sliding bubble area A_{sli} .

The nucleating stationary area of influence A_{sta} in pool boiling is defined based on the departure diameter D_d and the nucleate site density N_w :

$$A_{sta} = \min \left(A_{b,max}, K \left(\frac{\pi D_d^2}{4} \right) N_w \right) \quad (25)$$

The empirical constant K is typically set to 4 but can vary between 1.8 and 5. In the present study, the Del Valle and Kenning [39] correlation was used as follows:

$$K = 4.8e^{-\frac{Ja_{sub}}{80}} \quad (26)$$

where Ja_{sub} is the subcooled Jakob number.

In sliding conditions, the sliding bubbles' area of influence A_{sli} is taken into account:

$$A_{sli} = D_{avg} l_{sl} \left(\frac{\tau}{t_{sl}} \right) N_{ws} \quad (27)$$

where D_{avg} is the average bubble diameter during sliding, l_{sl} is the sliding distance, t_{sl} is the bubble sliding time, τ is time period at which transient conduction is dominant over convection, and N_{ws} is the revised nucleation site density due to sliding bubbles:

$$N_{ws} = \frac{1}{\left(\frac{l_{sl}}{g_s} \right) + 1} N_w \quad (28)$$

According to Basu [38], g_s represents the distance between two nucleation sites:

$$g_s = \frac{1}{\sqrt{N_w}} \quad (29)$$

The sliding bubble model accounts for bubble nucleation, growth, sliding, merging, and eventual lift-off along the heated wall (Figure 4).

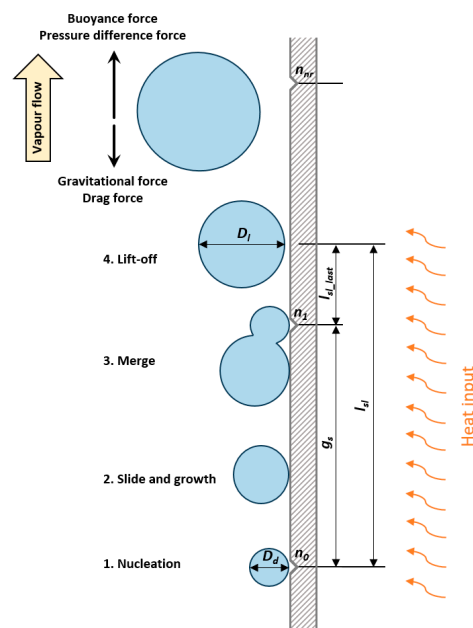


Figure 4. Sliding bubble.

Nucleated bubbles first grow and slide along the wall, during which they exchange heat with the liquid and contribute to the sliding heat flux. As they move, bubbles may

coalesce with neighbouring bubbles at a distance defined by the nucleation site spacing g_s , leading to accelerated growth. The sliding distance l_{sl} therefore depends on the number of merged bubbles n_{nr} and the remaining distance before lift-off [19,20], and is defined as

$$l_{sl} = g_s n_{nr} + l_{sl_last} \quad (30)$$

where n_{nr} represents the number of merged bubbles, and l_{sl_last} is the remaining sliding distance before the bubble lifts off.

Lift-off occurs once the bubble reaches the lift-off diameter D_l proposed by Shi and Zhang [20]:

$$D_l = 0.35 Ja_{sup}^{1.4} \exp(-0.114 Ja_{sub}) Re^{-0.36} lc \quad (31)$$

where lc is the capillary length which defines a length scale at which surface tension and buoyancy forces are in balance [22]. The corresponding sliding velocity is governed by the balance of buoyancy, pressure difference, gravitational and drag forces, and is written as

$$v_{sl} = \sqrt{\frac{2}{(C_D + \varepsilon_D) \rho_l} \left[\frac{4}{3} D_{avg} g (\rho_l - \rho_g) + \Delta P \right]} \quad (32)$$

The sliding velocity and lift-off diameter are obtained iteratively since the drag coefficient C_D depends on the bubble Reynolds number. The growth rate in the sliding process is calculated according to the method proposed by Gilman [19]:

$$D_{fra}^2 = D_{d+1}^2 - D_d^2 = \frac{t_s \alpha_l Ja_{sub}}{15 \left(0.015 + 0.0023 Re_b^{0.5} \right) \left(0.04 + 0.023 Ja_{sub}^{0.5} \right)} \quad (33)$$

where D_{fra}^2 is the evaporation growth fraction of the bubble after sliding for a time period of t_s . The increased bubble diameter during sliding D_{sli} is expressed as

$$D_{sli} = \sqrt{D_{fra}^2 + D_d^2} \quad (34)$$

If during the sliding the bubble reaches another nucleation site, it will absorb one or multiple bubbles before it lifts off. The absorption diameter can be written as [19]:

$$D_{n+1}^3 - D_n^3 = D_d^2 \quad (35)$$

where D_{n+1} and D_n are the diameters of a bubble before and after absorption, respectively.

Using the above formulations, the model explicitly captures the four key stages of bubble dynamics: (1) nucleation, (2) sliding and growth, (3) coalescence, and (4) lift-off.

2.7. Film Condensation Model

A simplified film condensation model based on Nusselt theory for laminar condensation on vertical walls [40] is used. Literature examples by Faghri [41] and Peterson [42] support the use of Nusselt-type laminar film correlations for TPCTs, and similar formulations have been adopted in recent TPCT studies (e.g., Chen [43]; Badache [44]).

The model assumes $T_w < T_{sat}$; a continuous liquid film is always present; liquid inertia is negligible; all wall heat transfer contributes to phase change. Saturation temperature is evaluated at the local cell pressure, $T_{sat} = T_{sat}(p)$.

The film thickness is approximated from the liquid holdup in each control volume:

$$h_l = \frac{\alpha_l V_P}{A_b} \quad (36)$$

where h_ℓ is film thickness; α_l is the liquid volume fraction; V_P is the cell volume; and A_b is the cell wall area exposed to condensation.

Assuming a linear temperature profile across the thin film (coordinate x normal to the wall), the temperature gradient across the film can be written as

$$\frac{\partial T}{\partial x} \cong \frac{(T_{sat} - T_w)}{h_l} \quad (37)$$

where, T_w is the wall temperature and T_{sat} the saturation temperature.

The wall heat flux is

$$\dot{q}_w = A_b k_l \frac{\partial T}{\partial x} \quad (38)$$

where k_l is the liquid thermal conductivity.

The corresponding cell condensation mass rate from an energy balance is

$$\dot{m}_{cond} = \frac{A_b k_l (T_{sat} - T_w)}{h_l h_{fg}} \quad (39)$$

where h_{fg} is the latent heat of vaporisation and h_l is the film thickness Equation (36).

2.8. Flow Evaporation and Condensation

Phase change in the bulk flow is modelled using the simplified Lee approach [45], which assumes local saturation-temperature equilibrium and uses empirical coefficients for calibration [45–47]. The corresponding mass source terms are

$$\Gamma_l = -\Gamma_v = f_c \alpha_v \rho_v \frac{(T - T_{sat})}{T_{sat}} \text{ for condensation } (T_v < T_{sat}), \quad (40)$$

$$\Gamma_v = -\Gamma_l = f_e \alpha_v \rho_v \frac{(T - T_{sat})}{T_{sat}} \text{ for evaporation } (T_l > T_{sat}). \quad (41)$$

Here, Γ_l and Γ_v are the mass source terms for the liquid and vapour phases, respectively; f_c and f_e are empirical condensation and evaporation coefficients; α_v is the vapour volume fractions; and ρ_v is the vapour density. T_{sat} is the saturation temperature (pressure-dependent), and T is the local mixture temperature.

The corresponding energy source terms are

$$H_c = \Gamma_M \Delta h \text{ for condensation, } (T < T_{sat}) \quad (42)$$

$$H_e = -\Gamma_M \Delta h \text{ for evaporation, } (T > T_{sat}) \quad (43)$$

where Γ_M is the phase-change mass flux and Δh the latent heat. The model is simplified in a way that the phase change occurs at the saturation temperature T_{sat} , and it uses empirical coefficients for its calibration. In present study, empirical coefficients f_c and f_e were set to 0.1, following recommendations from Lee [45] and subsequent studies [46,47].

2.9. Fluid–Solid Conjugate Heat Transfer

Conjugate heat transfer (CHT) between solid and fluid domains is modelled using an interfacial diffusion coefficient. The coefficient is derived from the heat flux $\dot{q}_w$, obtained using the modified RPI model, and the interface temperature difference $\Delta T_{int} = (T_w - T_l)$:

$$D = \frac{\dot{q}_w A}{\Delta T_{int} c_{p,l}} \quad (44)$$

where A is the heated surface area and $c_{p,l}$ the liquid-specific heat capacity.

In this implementation, only the sensible part of the RPI partition is used in the diffusion calculation (Equation (44)). This includes single-phase convection and quenching from stationary and sliding bubbles (Equation (21)). The resulting interfacial heat transfer coefficient is applied on the fluid side and the corresponding boundary condition is imposed on the solid side. The wall temperature T_w is recalculated in the solid, then passed back to the fluid. The procedure is iterated within each time-step until temperature and heat flux continuity are satisfied.

The latent part becomes active when the wall reaches T_{sat} and the evaporation partition of the RPI model is evaluated. The latent heat is taken from the solid as a heat sink equal to $\dot{m}_{eva}h_{fg}$. The corresponding vapour mass $\dot{m}_{eva}$ is added to the fluid at T_{sat} . During condensation at the wall, the latent heat is added to the solid as a heat source equal to $\dot{m}_{cond}h_{fg}$, and the fluid receives the corresponding liquid mass $\dot{m}_{cond}$ at T_{sat} . No separate latent term is added to the fluid energy equation because the enthalpy change is carried by the injected mass at T_{sat} together with the solid-side latent term, which prevents double counting and ensures energy conservation.

CBM CHT Boiling Compared to One-Sided Wall Boiling

We emphasise that the CBM model is, by design, a CHT model: it requires both solid and fluid domains with enforcement of interfacial continuity and is not applicable to one-sided wall boiling setups where the wall is a prescribed boundary.

In formulations without CHT, the interfacial thermal state is prescribed as either an isothermal T_w or an imposed wall heat flux $\dot{q}_w$. The RPI wall boiling model then operates on this fixed superheat or flux, so the predicted onset of nucleate boiling (ONB), the partitioned heat flux components, and the tendency toward dryout primarily reflect the chosen boundary condition.

With CHT enabled, the interface temperature and heat flux emerge from the coupled solid and fluid problem, using the interfacial diffusion coefficient (Equation (44)) together with the latent source and sink terms in the solid domain. The wall temperature T_w is constantly updated from the solved solid energy equation and fed back to the fluid; lateral conduction and wall heat capacity then redistribute energy, typically lowering and smoothing peak T_w (for imposed $\dot{q}_w$), shifting ONB timing/location, and altering the RPI partitioning. These effects can either stabilise or focus dry-spot growth depending on the solid wall configuration. Transiently, CHT damps the response, reducing overshoot in T_w .

In summary, including CHT by solving the solid energy equation and enforcing continuity of temperature and heat flux at the wall changes both the drivers of the wall boiling closures and the resulting fields in ways that a one-sided, non-CHT treatment cannot reproduce.

3. Computational Model

The updated Combined Boiling Model (CBM) simulations were compared against experimental data reported by Fadhl, Wrobel, and Jouhara [10] and detailed in Fadhl's dissertation [6]. The dataset provides thermal conditions in a TPCT at various heat inputs; in the present study, three representative operating conditions were selected. The experimental setup, including TPCT dimensions, is shown schematically in Figure 5.

Eight thermocouples were mounted on the outer wall of the heat pipe, with measurement positions distributed from the condenser to the evaporator; the detailed configuration is given by Fadhl [10]. The measured temperatures were compared with the simulated temperatures.

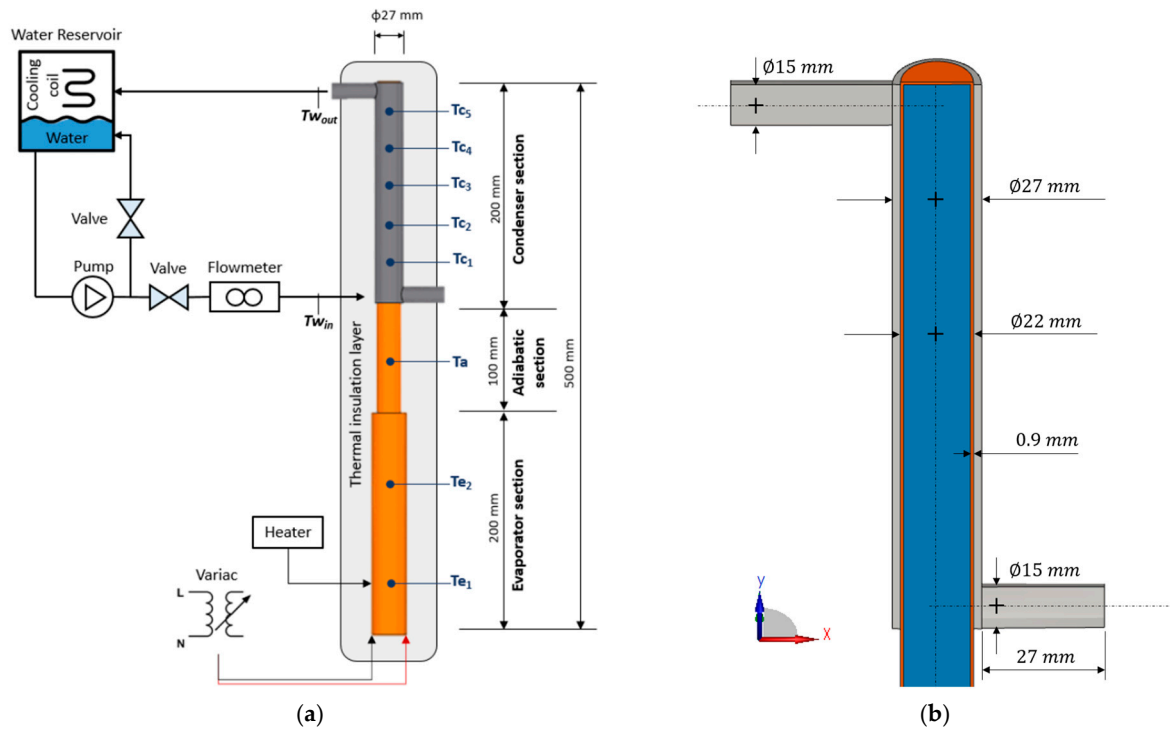


Figure 5. (a) Schematic representation of test rig; (b) TPCT condenser section dimensions.

3.1. Computational Mesh and Time-Step

Based on the test rig geometry, a multi-domain block-structured mesh was created, comprising 161,280 cells (Figure 6).

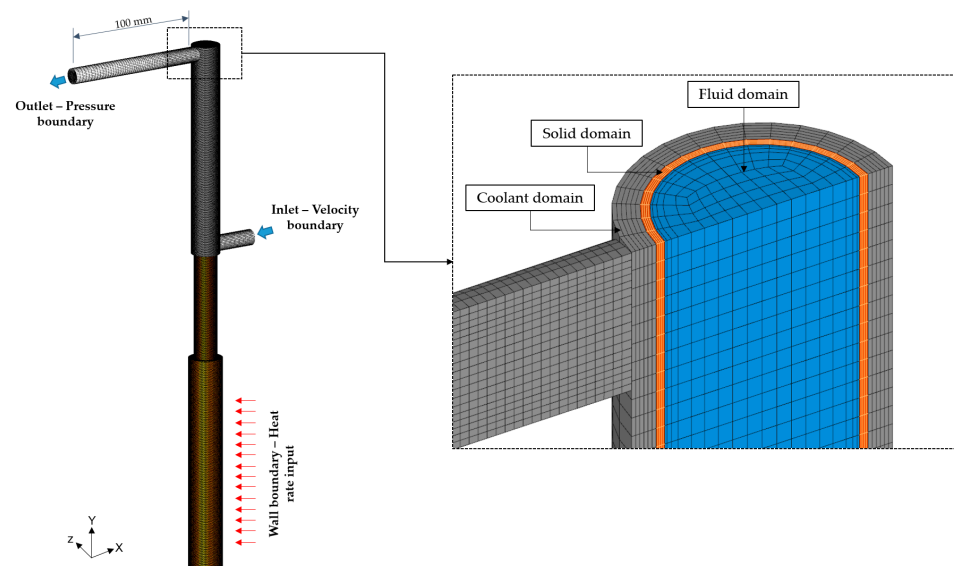


Figure 6. CFD block-structured mesh and boundary conditions.

The fluid domain contained 56,000 cells (blue), the solid copper domain 59,200 cells (orange), and the coolant domain 46,080 cells (grey).

A detailed sensitivity study for a similar CBM case was reported previously [22]; therefore, only a reduced mesh and time-step assessment was conducted here, guided by those conclusions. Meshes finer than 1 mm adequately resolved vapour distribution, with interface temperature deviations within ± 5 percent of experiments. Wall resolution yielded low y^+ values (8–20), which, although below the values often suggested for Euler–Euler

boiling simulations ($y^+ > 40$ [48]), ensured consistent temperature prediction. A 1.0 mm mesh with $\Delta t = 0.005$ s was identified as the optimal configuration.

In the present work, these settings were adopted with additional near-wall refinement (0.4 mm boundary layer and five solid layers of 0.18 mm each) to capture condensation films without artificial breakup in the Euler–Euler framework.

3.2. Numerical Settings

All simulations were performed in transient mode using the Euler–Euler framework with the models described above. Three computational domains were considered as follows: fluid, solid, and coolant. In the solid domain, only the energy equation was solved, while the fluid domain included continuity, momentum, turbulence, energy, and volume fraction equations. The Combined Boiling Model (CBM) was applied to capture near-wall phase change. Key simulation parameters for the multiphase fluid domain are summarised in Table 1.

Table 1. Transient simulation settings–multiphase fluid domain settings.

System Settings		
Run Mode	Transient	Time-Step: 0.005 s
Number of Iterations	Minimum: 5, Maximum 80	
Module	Multiphase:	Two Phases
Material	Liquid Phase: Water	Gas Phase: Vapour (Ideal Gas)
Activate Equations	Turbulence Model: Turbulence Wall Model: Energy: Wall Model:	k-ε-ζ-f Hybrid Static Enthalpy Standard
Additional Terms	Gravity:	9.81 m/s ²

The interfacial exchange models (Table 2) employed the CBM for mass and energy transfer, “Gas–Liquid System 3” for momentum exchange, Tomiyama’s correlation for bubbles (1 mm), and Schiller–Naumann for droplets (0.1 mm).

Table 2. Multiphase interfacial exchange settings used in the simulations.

Multiphase Interfacial Exchanges	CBM Model
Momentum Interfacial Exchange	“Gas–Liquid System 3” Gas (c)–Liq (d): Bubble Diameter = 1 mm (Tomiyama) Drop Diameter = 0.1 mm (Schiller–Naumann)
Mass and Energy Interfacial Exchange	CBM Model

Water properties were taken from the software database; vapour was modelled as a weakly compressible ideal gas. Phase change was restricted to the saturation temperature, with vapour pressure following the standard saturation curve (Figure 7). The saturation curve was taken from the built-in property database.

Solver stability and accuracy were maintained using the SIMPLE algorithm for pressure–velocity coupling. Convergence criteria were set to 1×10^{-4} for continuity, energy, and volume fraction, and 5×10^{-4} for momentum and turbulence. The MINMOD differencing scheme with 40% upwind blending [49] was used on momentum, volume fraction, and energy. Using these settings, the number of iterations fluctuated between 5 and 20 per time-step, with higher values occurring during boiling or condensation onset.

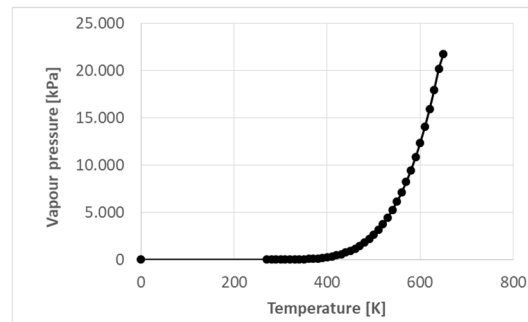


Figure 7. Vapour pressure curve.

3.3. Sensitivity Study

3.3.1. Sensitivity to Bubble and Droplet Departure Parameters

A sensitivity study on bubble and droplet departure parameters was partly carried out in the present work, because the values identified in our previous study, where the CBM was first introduced, were adopted as the baseline [22]. In that earlier study, the test case was a square channel (10×16 mm cross-section). Heat loads from 67 W to 836 W were applied as input, and a coolant inlet velocity from 0.25 m/s and 1 m/s. In the present study, the geometry is a circular tube with an inner diameter of 20.2 mm and input heat loads from 173 W to 376 W, while the bubble motion is driven by gravity and pressure differences within the TPCT. In terms of characteristic size and thermal loading, the two problems are therefore comparable, so using the previously obtained bubble and droplet parameters was a reasonable starting point.

It should be noted that the Schiller–Naumann [33] and Tomiyama [34] drag correlations used in the Euler–Euler two-fluid framework are empirical and semi-empirical models, respectively, and thus depend on the underlying experimental data. In the present configuration, they mainly affect vapour formation in the evaporation zone and the behaviour of the condensate film in the condenser. Since only the wall temperatures at selected TPCT locations were available for validation and these were reproduced satisfactorily, additional calibration of the interfacial models was considered unnecessary. Nevertheless, a reduced sensitivity check was performed to support future extensions of the model.

In the Euler–Euler framework, the chosen bubble/droplet diameter enters the interfacial momentum exchange: smaller diameters increase the interfacial area and thus the drag (i.e., the coupling between phases), whereas larger diameters reduce it. In the limiting case of very large diameters, the phases approach a decoupled behaviour; in the opposite limit of very small diameters, the flow tends toward a nearly homogeneous mixture. From a numerical point of view, weak phase coupling (too large diameters) generally deteriorates convergence, so unrealistically large values are not recommended.

The values used in the investigation were 1 mm for the bubble diameter and 0.1 mm for the droplet diameter, which is in line with values reported in the literature.

For example, Kocamustafaogullari [50] reports bubble diameters from about 0.2 mm up to 5 mm depending on the operating conditions. Wang [51] in his work on a flat two-phase thermosiphon shows individual bubbles of a few millimetres before lift-off, and Li [52], in his recent MDPI visual study, shows evaporator frames where discrete bubbles detach and form vapour plugs again in the millimetre range.

The droplet diameter of 0.1 mm is more difficult to justify directly, because in the present case, we are not only modelling small dispersed droplets in the vapour core, but mainly the wall condensate film, which is, in reality, continuous. To mimic such wall film in the Euler–Euler model, the droplet diameter was taken to be of the same order as the expected film thickness, so that the liquid phase stays attached to the wall, has enough

momentum to drain downward under gravity, and is not swept away by the vapour drag before a stable laminar film forms. According to the examples given by Faghri [41] and Peterson [42], the condensate in the condenser of a wickless heat pipe (TPCT) can be treated as a Nusselt-type laminar film and predicted with the classical vertical-plate solution [40]. For water at typical operating conditions, this gives local film thicknesses of about 20–80 μm , increasing slowly with distance from the top, so 0.1 mm (100 μm) was adopted as an appropriate and slightly conservative modelling parameter.

For testing the condensing film and the influence of droplet size, a reduced model was built. The model included the heat exchanger (cooler) and the TPCT, using the same diameter and cell distribution as the full TPCT. It was initialised with pure vapour phase, and no external heating was applied. As the walls were cooled, condensate formed on the colder surface, the pressure decreased, and a circulating vapour flow developed within the reduced TPCT. The test setup and results are shown in Figure 8.

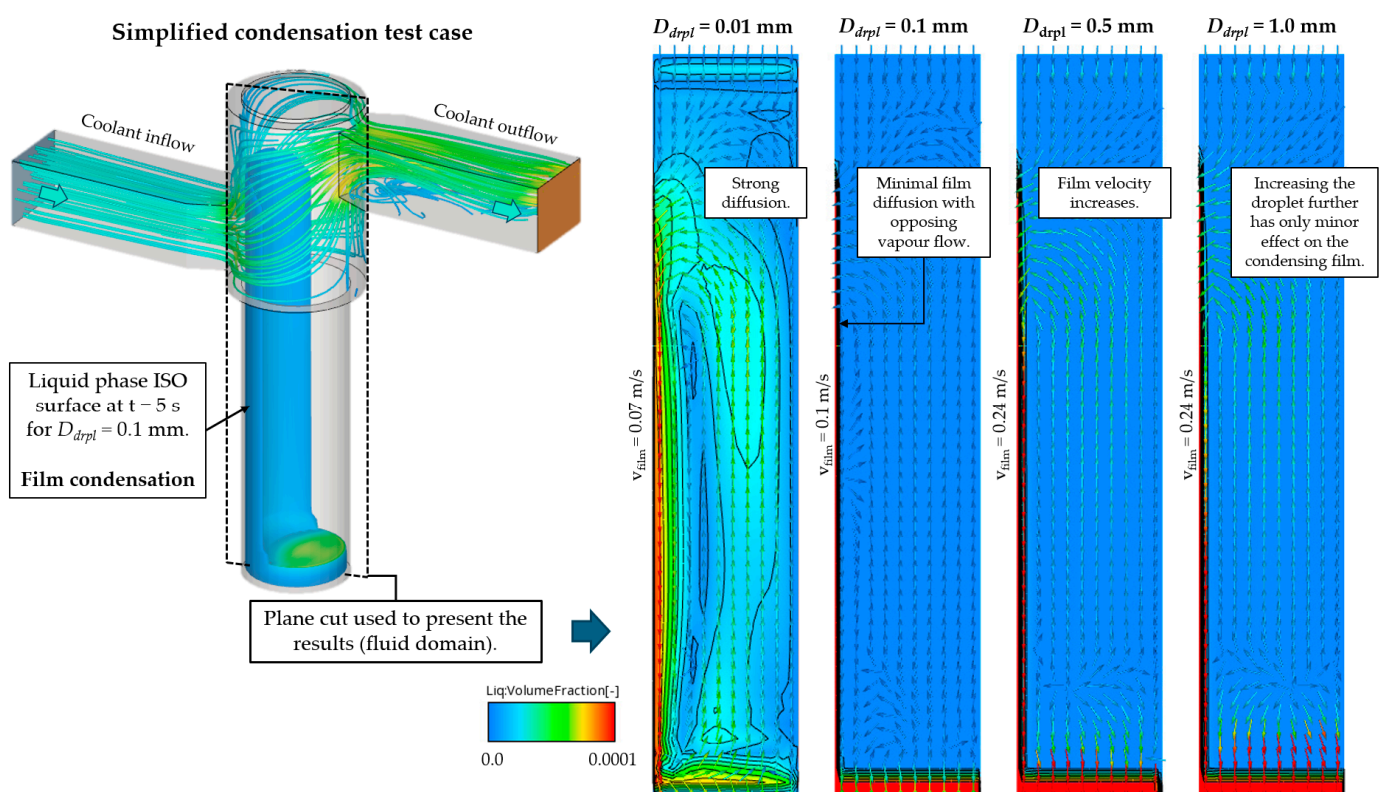


Figure 8. Droplet-size sensitivity test case and resulting volume fraction distribution.

Figure 8 shows film formation induced by wall cooling at four droplet sizes. A droplet diameter of 0.01 mm proved too small, leading to overly strong phase coupling and excessive interphase diffusion. A diameter of 0.1 mm produced the weakest observed film diffusion under counter-current vapour flow and the most numerically robust, converging consistently in the fewest number of iterations. Increasing the diameter to 0.5 mm and 1.0 mm reduced phase coupling, which increased the liquid-phase velocity and caused a slight rise in diffusion, together with somewhat reduced numerical stability.

To assess the influence of bubble diameter on the flow, the earlier simplified model was extended in length and an evaporation section was added, yielding a full TPCT geometry. This test geometry closely resembled the final model but was shortened to 325 mm to accelerate the study. The diameter and cell distribution were identical to those of the final TPCT. The reduced TPCT model was also used for the mesh and time-step sensitivity analyses.

The reduced TPCT test case and the droplet-diameter sensitivity results are presented in Figure 9. Five bubble diameters were examined (0.1–5 mm); however, the 0.1 mm case was numerically unstable and is therefore omitted. At 0.5 mm, a slightly stronger liquid-phase diffusivity was observed; otherwise, the results were very similar. The thermal metrics differed by less than 1% across the remaining cases (0.5–5 mm), indicating that, for this application, bubble size within the tested range has only a minor influence on the results.

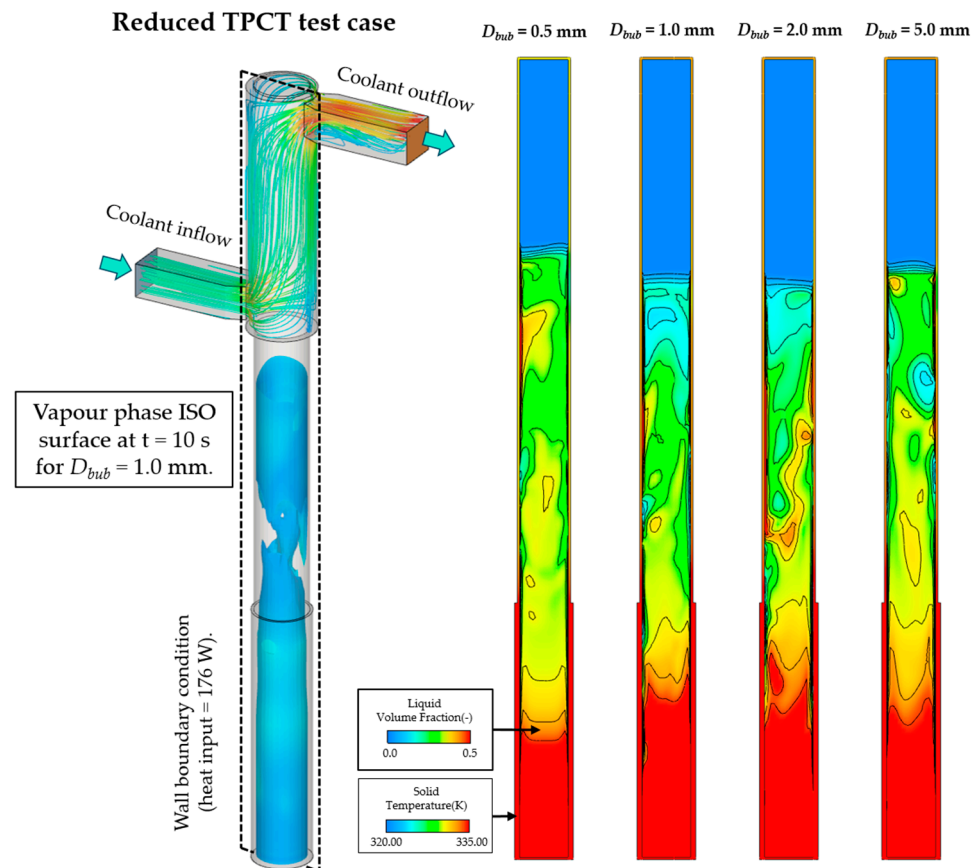


Figure 9. Bubble-size sensitivity test case and resulting volume fraction and solid temperature distribution.

3.3.2. Mesh and Time-Step Sensitivity Study

As a starting point, the TPCT cell size applied followed the recommendations of previous work [22]. This baseline mesh was then compared with coarser and finer meshes. Temperature at three locations on the TPCT solid walls were monitored and compared between each other. The monitoring locations and the meshes used in the study are shown in Figure 10.

Key mesh features all listed in Table 3.

Table 4 reports temperatures at three key monitoring positions within the TPCT solid domain. Results from Mesh2 and Mesh3 are essentially the same, while Mesh1 shows deviations of up to 8 K in the adiabatic and evaporator sections relative to Mesh2/3.

The differences between the cases are further illustrated by temperature and volume fraction cut-plane plots in Figure 11. Mesh1 shows greater numerical diffusion, yielding a lower liquid temperature compared with Mesh2 and Mesh3. Mesh2 was selected for subsequent simulations because it produced results comparable to the more expensive Mesh3 at lower cost. This outcome is consistent with our previous research findings [22].

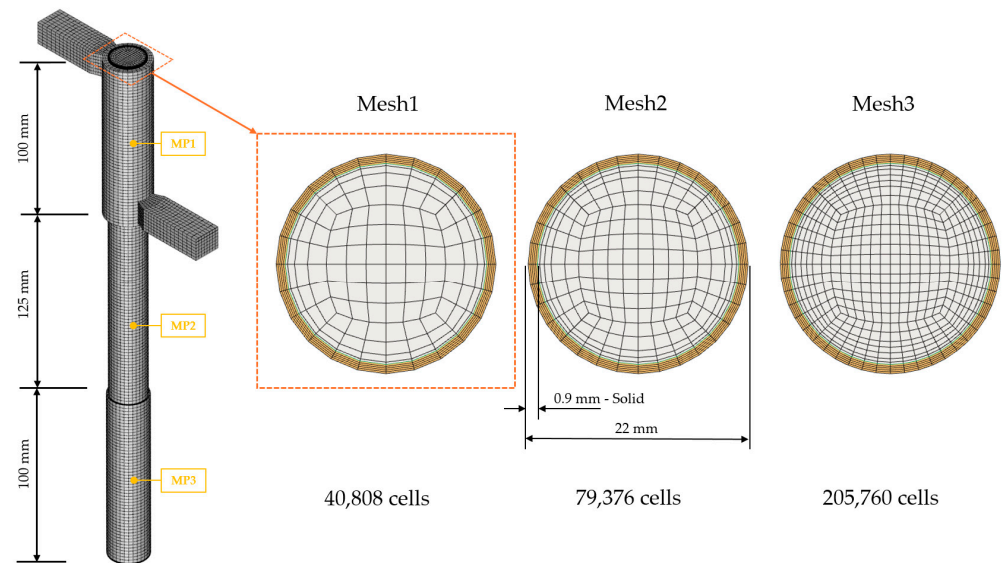


Figure 10. Meshes used in the mesh sensitivity study.

Table 3. Mesh resolution employed in the mesh sensitivity study.

Mesh Name	Cell Size [mm]	Total Number of Cells	Boundary Layer Thickness [mm]	Wall y^+ Range
Mesh1	1–2	40,808	0.5	0–21
Mesh2	0.5–1.5	79,376	0.35	0–28
Mesh3	0.4–1.0	205,760	0.3	0–41

Wall y^+ range is shown for liquid phase at $t = 30$ s.

Table 4. Temperature at monitoring points in the mesh sensitivity study at $t = 30$ s.

Section	Monitoring Position	Mesh1 [K]	Mesh2 [K]	Mesh3 [K]
Evaporator	MP3	344.14	343.50	343.55
Adiabatic	MP2	332.84	339.67	340.08
Condenser	MP1	330.00	338.30	338.42
Relative Computational Cost [%]		100	220	700

To select an appropriate time-step, the effect of varying time-step size on the solution was assessed. Because an Euler–Euler semi-implicit time-marching scheme is employed, simulations can be advanced with a Courant–Friedrichs–Lewy (CFL) number [53] greater than 1. Five time-steps were evaluated, and the resulting fields and temperatures at key locations were analysed. The results are summarised in Table 5.

Table 5. Tested time-step values, corresponding maximum CFL numbers, temperatures at monitoring positions, and computational cost at $t = 30$ s.

Simulation Time-Step [s]	CFL Liquid [-]	CFL Vapour [-]	MP1 [K]	MP2 [K]	MP3 [K]	Max Deviation from Ref. Case APE [%]	Relative Comp. Cost [%]
0.025	9.03	41.08	335.85	346.65	351.71	2.6	100
0.01	3.246	15.65	343.00	343.57	348.42	3.5	76
0.005	2.233	12.19	337.64	339.16	343.78	1.25	112
0.0025	1.11	9.17	334.10	339.51	339.51	0.01	162
0.001	0.52	4.68	334.05	339.48	339.48	Reference	290

The maximum deviation was computed relative to the finest case ($\Delta t = 0.001$ s).

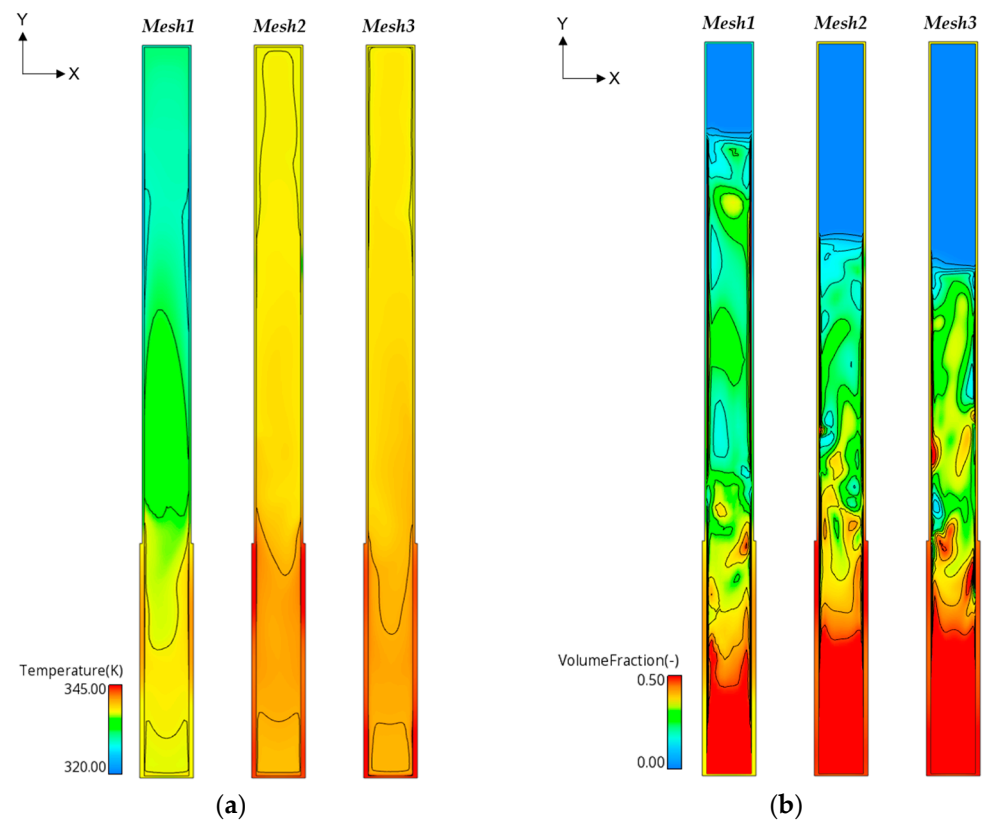


Figure 11. (a) Liquid temperature distribution at $t = 30$ s for Mesh1, Mesh2, and Mesh3; (b) liquid volume fraction distribution at $t = 30$ s for Mesh1, Mesh2, and Mesh3.

Although the largest time-step of 0.025 s showed better agreement than 0.01 s, pronounced temperature artefacts were visible in the temperature field, so this step was deemed unsuitable. A time-step of 0.01 s is the largest plausible choice; its largest difference from the finer solution was $\sim 3.5\%$. The 0.005 s step provided the best compromise between computational cost and deviation and was therefore selected for further simulations. For time-steps of 0.0025 s and smaller, the results changed little, remaining essentially unchanged despite further reductions. These observations agree with our earlier study [22].

3.4. Initial and Boundary Conditions

Boundary conditions were specified for all three computational domains. In the condenser heat exchanger domain, a water inlet mass flow rate of 0.003 kg/s at 293.75 K was imposed, while the outlet was set to 1 bar (absolute) (Figure 6). The external walls were treated as adiabatic. The solid domain was defined as copper, and a prescribed heat rate boundary condition was applied at the evaporator section (Figure 6). Five heat load cases were simulated as follows: 173 W and 376 W with a filling ratio of 1.0, and 173 W, 225 W, and 376 W with a filling ratio of 0.5. The filling ratio denotes the fraction of the evaporator volume initially occupied by liquid (e.g., 1.0 for a fully liquid-filled evaporator, 0.5 for a half-filled evaporator).

4. Results and Discussion

4.1. Initial Flow Simulation

The simulations were conducted in two stages. In the first stage, the coolant flow field was solved until a quasi-steady state was reached. This solution was subsequently used to initialise the full conjugate heat transfer (CHT) model, thereby ensuring a physically consistent start-up condition for the coupled phase-change simulation.

The computed coolant velocity field is illustrated in Figure 12a, where the colour contours represent the velocity magnitude and the streamlines indicate the flow direction. The coolant flow simulation was performed for a total duration of 60 s; however, a quasi-steady state was reached after approximately 30 s, as confirmed by the outlet mass flow rate presented in Figure 12b.

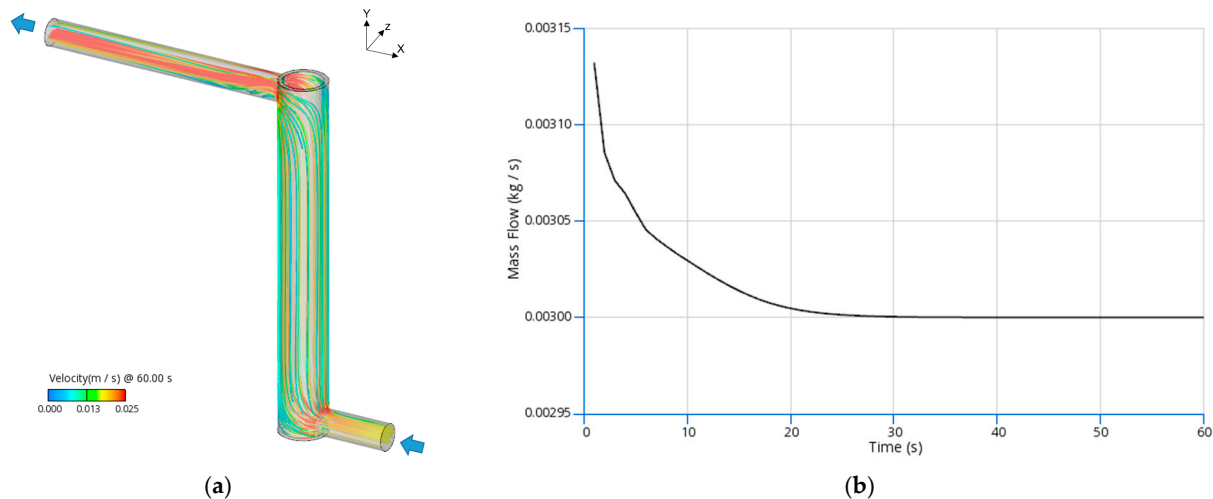


Figure 12. (a) Steady coolant flow; (b) outlet mass flow rate.

This preliminary step provided a stable and realistic hydrodynamic field for the subsequent CHT simulations, minimizing numerical transients during model initialisation.

4.2. CHT Full-Model Simulation

In the present preliminary validation study, five CHT simulations were performed and analysed, of which three were compared directly with available experimental and numerical data [10]. The coupled vapour–liquid flow dynamics developed primarily during the initial transient period, while the overall thermal response required a longer time to stabilise.

The transient development of the liquid and vapour phases is best observed for the filling ratio 1.0, in which the entire evaporator section was initially filled with liquid water, and the remaining adiabatic and condenser sections contained pure vapour. The evolution of the liquid volume fraction for two representative heat loads (173 W and 376 W) is shown in Figure 13.

At the beginning of the simulation, a rapid evaporation rate was observed in both cases, stabilizing after approximately 10 s. The larger vapour generation at higher heat input (376 W) became evident after this period. The formation of large vapour pockets was observed in the adiabatic section before collapsing into the condenser region. Within the condenser, the simulations consistently predicted the formation of a thin liquid film on the upper wall surface, as illustrated in Figure 14.

The fluid flow in this region clearly shows vapour transport toward the upper condensation surface, followed by flow reversal and downward liquid film drainage. This behaviour is consistent with experimental observations and confirms that the present model is capable of capturing the essential evaporation–condensation mechanisms occurring within the TPCT.

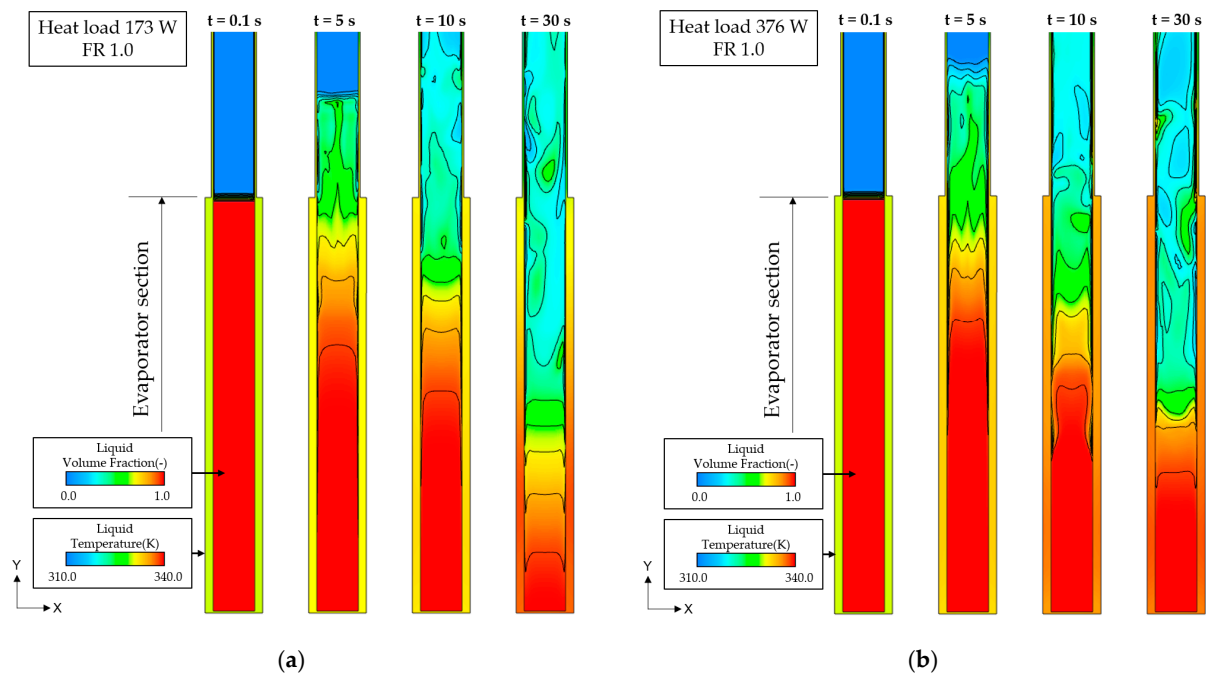


Figure 13. (a) Water volume fraction distribution at 0.1 s, 2.5 s, 5 s, and 10 s for 173 W; (b) water volume fraction distribution at 0.1 s, 2.5 s, 5 s, and 10 s for 376 W.

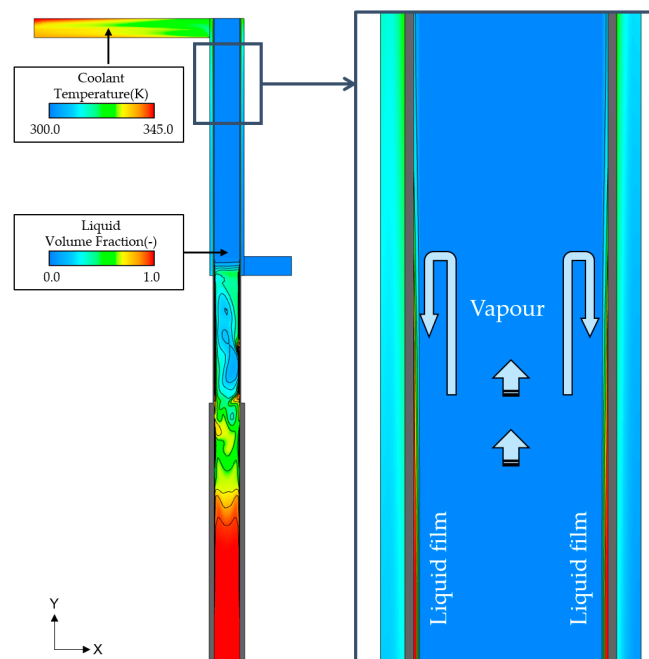


Figure 14. Predicted condensing film formation.

4.3. Preliminary Model Validation

Experimental data used for the preliminary validation were available only for a TPCT with a filling ratio of 0.5 [6]. Three heat input cases were investigated as follows: 173 W, 225 W, and 376 W.

With less liquid in the evaporator section, the TPCT's liquid flow behaviour differs from the case with the filling ratio of 1.0. Figure 15 shows the flow evolution during the initial transient period for the two representative heat inputs (173 W and 376 W). In the filling ratio 1.0 case, mixed two-phase flow develops gradually toward the condenser whereas, in the filling ratio 0.5 case, a sudden two-phase front appears at about 4 s and then

slowly settles. At approximately 30 s (near steady state), a pure liquid region remains only in roughly the first quarter of the evaporator; the rest of the liquid circulates as a two-phase mixture in the pipe.

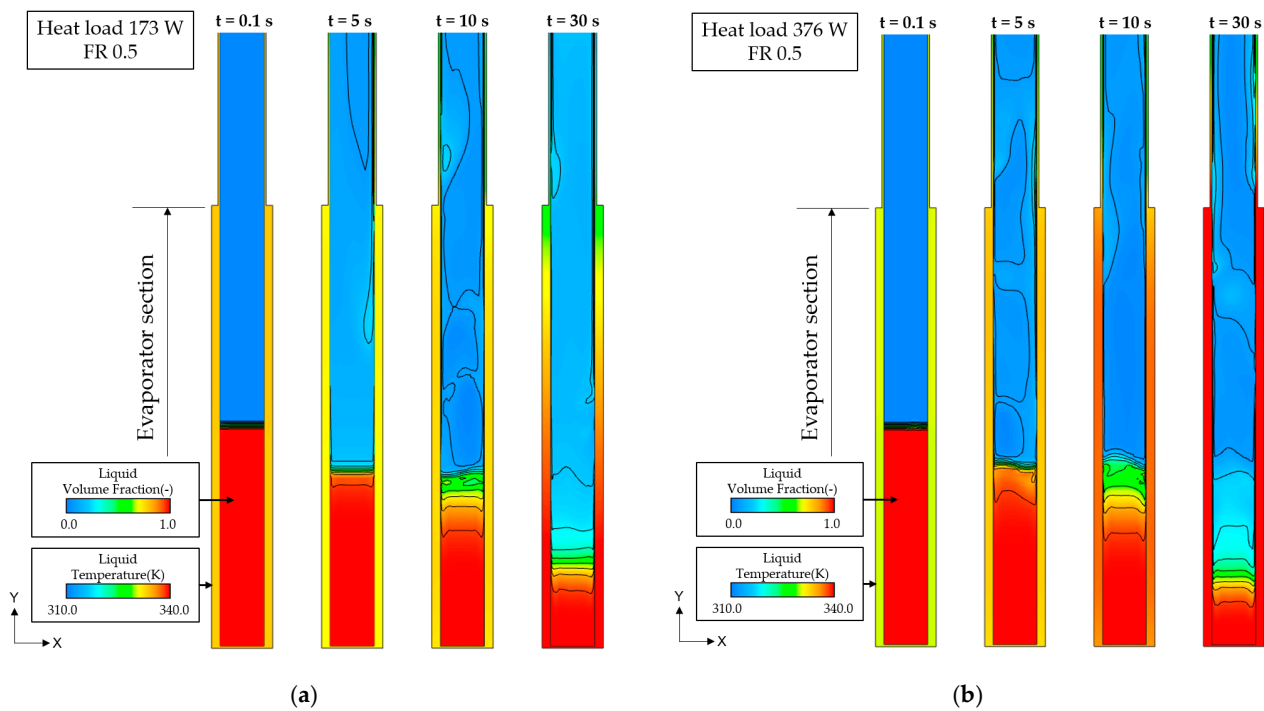


Figure 15. (a) Heat load 173 W water volume fraction distribution at 0 s, 5 s, 10 s, and 30 s; (b) heat load 376 W water volume fraction distribution at 0 s, 5 s, 10 s, and 30 s.

Comparing the 173 W and 376 W heat input cases at a filling ratio of 0.5, the start-up phase (up to ~8 s) shows similar evaporation patterns within the evaporator for both cases. However, after a time period of 30 s, the 376 W case displayed significantly less residual liquid in the evaporator section and a noticeably thicker condensate film in the condenser section. The stronger evaporation and higher vapour production observed at 376 W are consistent with the expected temperature increase under higher thermal loading. The influence of heat input on the overall temperature distribution at quasi-steady state is shown in Figure 16.

Pronounced temperature gradients are observed across the TPCT, with the maximum temperatures located in the evaporator at 376 W. A modest flow asymmetry was observed, attributable to the lower position of the coolant inlet. Colder coolant entered from the bottom, absorbed heat as it travelled through the condenser, and gradually warmed before exiting at the opposite upper outlet.

Simulations were also evaluated quantitatively. Final temperature predictions at eight solid locations (Figure 5) were compared directly with experimental measurements and with prior numerical results obtained using the VOF–Lee model [8].

Absolute percent error (%) with respect to the experimental value, used in the following comparisons, was computed as

$$APE = \left| \frac{T_{CFD} - T_{EXP}}{T_{EXP}} \right| \times 100\% \quad (45)$$

Here, T_{CFD} is the simulated temperature at the observation point, T_{EXP} is the measured temperature at the same location, and APE denotes the absolute percent error.

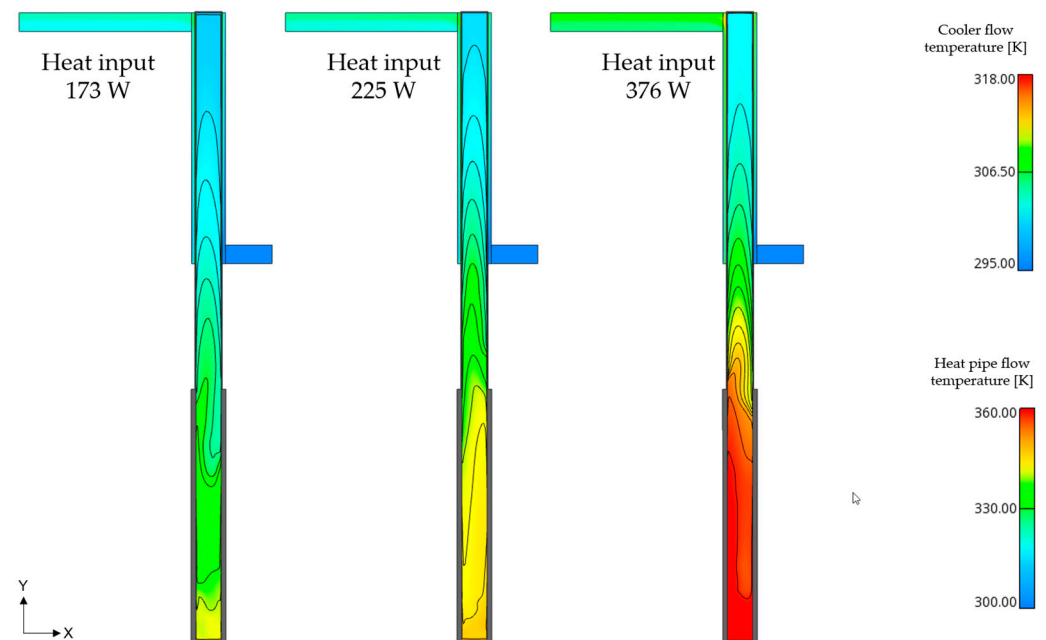


Figure 16. Temperature distribution at $t = 60$ s for heat inputs of 173, 225, and 376 W.

The comparison for a 173 W heat input at $t = 60$ s is presented in Table 6. The results show that the APE in the condensation section of the TPCT remained below 3.5%, despite the use of a simplified condensation model. The largest improvement over the VOF–Lee results occurs in the evaporation section, where APEs were reduced to 1.27% and 1.79% (previously 9.42% and 12.14%). In the adiabatic section, the APE decreased from 10.68% (VOF–Lee) to 3.13% (CBM). Moreover, the spatial temperature distribution is reproduced without the over-prediction bias observed in the VOF–Lee solution, suggesting improved robustness under near-quasi-steady conditions.

Table 6. Comparison between experimental data, previous numerical results, and current CFD predictions at $t = 60$ s for 173 W (filling ratio 0.5).

Section	Monitoring Position	T_{EXP} [K]	T_{CFD} [K]	APE [%]	T_{CFD} [K]	APE [%]
			Lee Model	Lee Model	CBM Model	CBM Model
Evaporator	Te ₁	345.75	378.33	9.42	339.57	1.79
	Te ₂	337.45	378.40	12.14	333.16	1.27
Adiabatic	Ta	327.45	362.41	10.68	317.20	3.13
Condenser	Tc ₁	320.55	329.54	2.80	309.33	3.50
	Tc ₂	318.85	326.54	2.41	310.26	2.69
	Tc ₃	317.95	325.95	2.52	310.63	2.30
	Tc ₄	317.09	325.64	2.71	310.95	1.94
	Tc ₅	315.95	327.13	3.54	311.17	1.51

Experimental data and CFD Lee model simulation results from Fadhl [6].

The comparison for a 225 W heat input at $t = 60$ s is presented in Table 7. The VOF–Lee model shows slightly lower errors in the condensation section, with APE typically below 1%, while the CBM exhibits APEs between 2% and 4% in this region. In the evaporation and adiabatic sections, the CBM yields lower APEs of 2.30%, 1.66%, and 3.38%, compared with 2.72%, 10.49%, and 10.32% for the VOF–Lee model. Thus, the CBM provides smaller errors where thermal gradients are strongest, with a modest increase in the condensation-section error.

Table 7. Comparison between experimental data, previous numerical results, and current CFD predictions at $t = 60$ s for 225 W (filling ratio 0.5).

Section	Monitoring Position	T_{EXP} [K]	T_{CFD} [K]	APE [%]	T_{CFD} [K]	APE [%]
			Lee Model	Lee Model	CBM Model	CBM Model
Evaporator	Te ₁	352.68	379.91	2.72	344.57	2.30
	Te ₂	343.41	379.44	10.49	337.7	1.66
Adiabatic	Ta	330.98	365.13	10.32	319.79	3.38
Condenser	Tc ₁	322.93	326.01	0.95	309.72	4.09
	Tc ₂	320.24	323.15	0.91	311.38	2.77
	Tc ₃	321.22	322.44	0.38	311.33	3.08
	Tc ₄	319.51	322.20	0.84	311.73	2.43
	Tc ₅	318.29	322.67	1.38	311.82	2.03

Experimental data and CFD Lee model simulation results from Fadhl [6].

The comparison for a 376 W heat input at $t = 60$ s is presented in Table 8. The CBM yields lower errors in the evaporation and adiabatic sections, with APEs between 1.9% and 3.5%, while slightly larger deviations are observed in the condensation section (up to 5.09%). In that region, the VOF–Lee simulation gives marginally closer results, with a maximum APE of 3.05%.

Table 8. Comparison between experimental data, previous numerical results, and current CFD predictions at $t = 60$ s for 376 W (filling ratio 0.5).

Section	Monitoring Position	T_{EXP} [K]	T_{CFD} [K]	APE [%]	T_{CFD} [K]	APE [%]
			Lee Model	Lee Model	CBM Model	CBM Model
Evaporator	Te ₁	376.75	385.14	2.23	369.02	2.05
	Te ₂	363.65	384.97	5.86	356.73	1.90
Adiabatic	Ta	342.75	370.11	7.98	330.74	3.50
Condenser	Tc ₁	328.95	327.12	0.56	316.34	3.83
	Tc ₂	325.55	323.66	0.58	317.00	2.63
	Tc ₃	332.45	323.15	2.80	316.73	4.73
	Tc ₄	331.35	322.70	2.61	316.18	4.58
	Tc ₅	333.35	323.17	3.05	316.37	5.09

Experimental data and CFD Lee model simulation results from Fadhl [6].

4.4. Overall Model Performance

The overall model performance, together with its comparison against experimental data and the Lee model results, is presented in Figure 17. Although detailed deviations from the experimental measurements were provided in the previous tables, the overall agreement can be more clearly visualised by examining the trend line curves. The markers in the figure correspond to the individual measurement points, while the solid lines represent the fitted trend lines. From the results, it can be concluded that the proposed CBM provides a markedly improved prediction of the thermal behaviour of the analysed TPCT across all investigated heat loads.

While the condensation sub-model integrated within the CBM framework remains simplified, the achieved level of agreement with experimental data indicates that the CBM approach successfully captures the dominant physical mechanisms governing evaporation–condensation interactions. Further refinement of the condensation treatment is expected to reduce residual deviations, particularly under varying thermal loads and transient operating conditions.

In summary, the proposed model demonstrates robust predictive capability for coupled phase-change dynamics in TPCT and provides a solid foundation for the future development of high-fidelity, physics-based phase-change models in CFD.

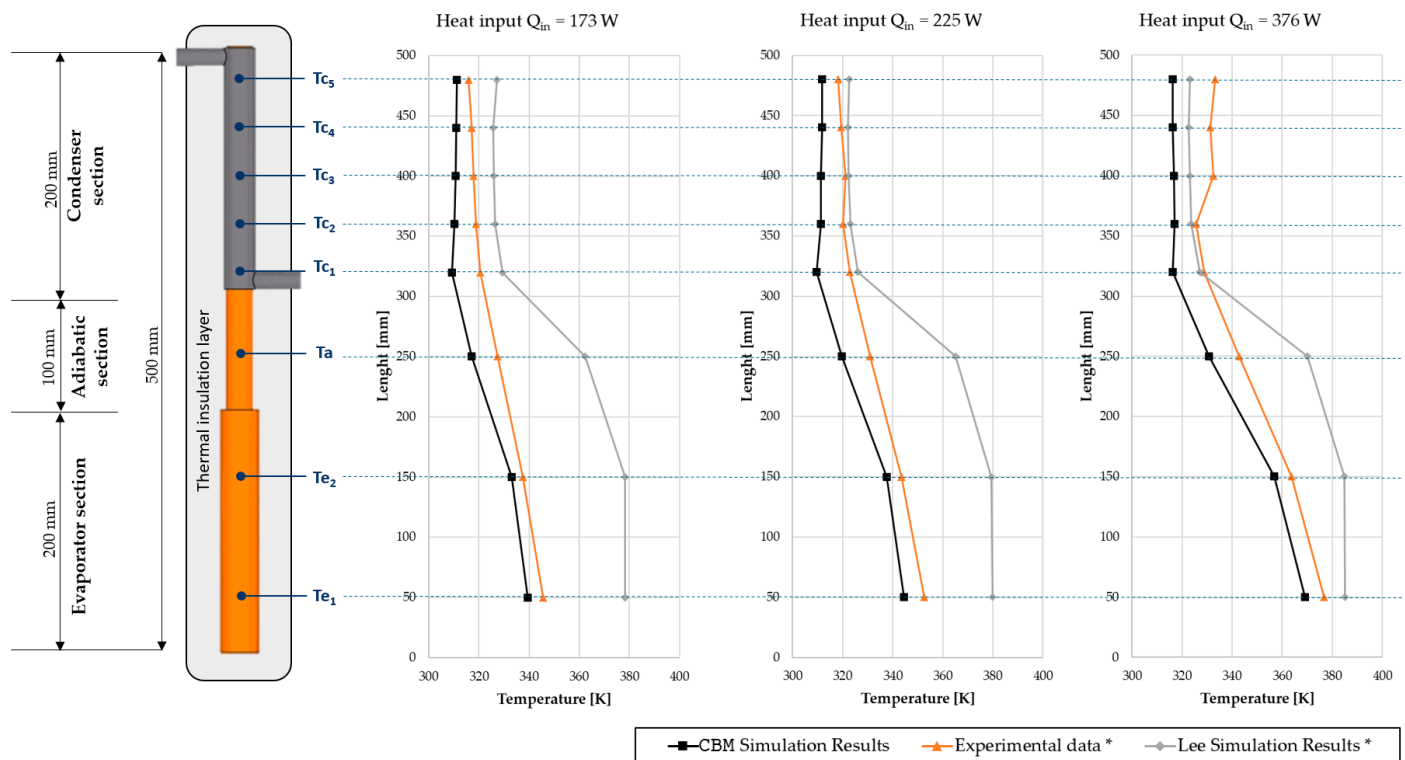


Figure 17. Comparison of temperature distributions at various heat inputs. An asterisk (*) indicates data taken from Fadhl [6] (experimental data and Lee’s simulation results).

5. Conclusions

This work presented a CFD implementation and preliminary validation of a Combined Boiling Model (CBM) for a two-phase closed thermosiphon (TPCT) within a conjugate heat transfer framework. The model integrates a modified RPI wall boiling formulation, film condensation, and bulk phase change in an Euler–Euler formulation, and was assessed against experiments for heat inputs from 173 W to 376 W. The simulations reproduced the principal thermal and flow features of a TPCT, including vapour generation in the evaporator, vapour-pocket formation and collapse in the adiabatic section, and thin-film condensation on the condenser wall. Predicted wall temperatures agreed well with measurements across the tested operating points, with the strongest accuracy in the evaporator and adiabatic regions and competitive performance in the condenser despite a simplified condensation model. Compared with a representative VOF–Lee formulation, the CBM achieved similar or lower temperature deviations where thermal gradients are largest while allowing substantially larger time-steps and coarser meshes, which lowers computational cost and makes the approach suitable for system-level thermal analysis in engineering workflows.

In terms of computational effort, fully three-dimensional VOF–Lee simulations in the literature required very long runtimes. For example, in Fadhl’s study, approximately two weeks of wall time were needed to simulate about 2 s of thermosyphon operation [6], which limited the number of operating points investigated. In the present work, TPCT simulations using the CBM reached 60 s of physical time in about 4 days for filling ratio 0.5 and about 6 days for filling ratio 1.0. These timings are indicative and depend on hardware and parallelisation. Our runs used 40 CPUs (2×20 cores) on Intel E5-2690 v4 with 128 GB RAM.

Several limitations follow from the modelling choices and indicate directions for future work. The CBM employs empirical and semi-empirical correlations, including nucleation site density, bubble departure size and frequency, interfacial heat transfer, and interfacial momentum closures. These may require recalibration when the geometry, fluid, surface condition, or operating range changes. Local thermal equilibrium is enforced within the interfacial layers used to evaluate phase-change sources, which is appropriate at engineering resolution but can suppress fine-scale non-equilibrium effects near the wall. The Euler–Euler framework does not explicitly resolve the liquid–vapour interface, so interface-controlled phenomena that depend on detailed interface topology, for example geyser boiling, capillary driven microscale dynamics, thin-film rupture and rewetting, and certain interfacial instabilities that are naturally captured by VOF, are outside the high-fidelity scope of the present model. These trade-offs are consistent with the intended use. The CBM offers a physically consistent and computationally efficient tool for predicting temperature fields and overall heat transport in two-phase devices, which is the primary objective in engineering thermal analysis and design.

Future work will extend validation to broader operating ranges and working fluids, refine condensation and interfacial heat transfer closures, assess sensitivity of calibrated coefficients across surfaces and fill ratios, and selectively couple to interface resolving methods where microscale effects dominate. Within these bounds, the proposed CBM with CHT provides a robust, practical, and accurate approach for the thermal analysis of thermosyphons and related heat pipe systems.

Author Contributions: Conceptualization, J.Š. and M.Z.; methodology, J.Š. and M.Z.; software, J.Š.; validation, J.Š.; formal analysis, J.Š.; investigation, J.Š.; resources, J.Š. and M.Z.; data curation, J.Š. and M.Z.; writing—original draft preparation, J.Š., M.Z. and J.M.; writing—review and editing, J.Š., M.Z. and J.M.; visualization, J.Š.; supervision, M.Z. and J.M.; project administration, M.Z.; funding acquisition, M.Z. and J.M. All authors have read and agreed to the published version of the manuscript.

Funding: This research received no external funding.

Data Availability Statement: Data are available on reasonable request from the corresponding author. Portions of the dataset are subject to third-party intellectual-property and confidentiality obligations and cannot be disclosed. Only materials cleared for release (non-confidential and non-proprietary) can be shared.

Acknowledgments: The study was undertaken under the guidance and support of the Faculty of Mechanical Engineering, University of Maribor (Chair of Power, Process, and Environmental Engineering). The authors further acknowledge the technical support of AVL-AST d.o.o. The authors wish to thank the Slovenian Research and Innovation Agency (ARIS) for the financial support in the framework of the Programme P2-0196: Research in Power, Process and Environmental Engineering.

Conflicts of Interest: Author Jure Štruel was employed by the company AVL-AST d.o.o. The remaining authors declare that the research was conducted in the absence of any commercial or financial relationships that could be construed as a potential conflict of interest.

Abbreviations

The following abbreviations are used in this manuscript:

CBM	Combined Boiling Model
CHT	Conjugate Heat Transfer
CFD	Computational Fluid Dynamics
TPCT	Two-Phase Closed Thermosyphon
VOF	Volume of Fluid

RPI	(Kurul–Podowski) wall boiling model, commonly called the RPI model
SIMPLE	Semi-Implicit Method for Pressure-Linked Equations (pressure–velocity coupling)
$k-\varepsilon-\zeta-f$	Turbulence model (k-epsilon-zeta-f)
ONB	Onset of Nucleate Boiling
LTE	Local Thermal Equilibrium
APE	Absolute Percent Error
CFL	Courant–Friedrichs–Lewy number
FR	Filling ratio

Nomenclature

Symbol	Description	Unit
A	Area (generic, e.g., heated surface)	m^2
A_b	Boiling area fraction	–
a_i	Interfacial area density	m^{-1}
c_p	Specific heat capacity	$J \cdot kg^{-1} \cdot K^{-1}$
C_D	Drag coefficient	–
D	Diameter (generic)	m
d_{32}	Sauter mean diameter	m
Eo	Eötvös number	–
F	Bubble departure frequency	s^{-1}
F_D	Interfacial drag force (per unit volume)	$N \cdot m^{-3}$
G	Gravitational acceleration	$m \cdot s^{-2}$
g_s	Nucleation site spacing	m
H	Specific enthalpy	$J \cdot kg^{-1}$
h_i	Interfacial heat transfer coefficient	$W \cdot m^{-2} \cdot K^{-1}$
h_{fg}	Latent heat of vaporisation	$J \cdot kg^{-1}$
Ja	Jakob number	–
K	Thermal conductivity	$W \cdot m^{-1} \cdot K^{-1}$
l_{sl}	Sliding distance	m
l_c	Capillary length	m
M	Interfacial momentum exchange source (per unit volume)	$N \cdot m^{-3}$
$\dot{m}$	Mass rate (context-dependent: per area or per volume)	$kg \cdot s^{-1}$
N_w	Nucleation site density	m^{-2}
Nu	Nusselt number	–
P	Pressure	Pa
Pr	Prandtl number	–
$\dot{q}$	Heat flux	$W \cdot m^{-2}$
Re	Reynolds number	–
t_{sl}	Bubble sliding time	s
T	Temperature	K
V_P	Cell (control-volume) volume	m^3
$\mathbf{v}$	Velocity (vector)	$m \cdot s^{-1}$
$\mathbf{v}_r$	Slip velocity magnitude ($ \mathbf{v}_d - \mathbf{v}_c $)	$m \cdot s^{-1}$
ΔP	Pressure difference	Pa
Δh	Latent enthalpy change	$J \cdot kg^{-1}$
ΔT	Temperature difference (generic)	K
Γ	Mass transfer rate (per unit volume)	$kg \cdot m^{-3} \cdot s^{-1}$
Greek letters		
α	Volume fraction	–
μ	Dynamic viscosity	$Pa \cdot s$
ρ	Density	$kg \cdot m^{-3}$
σ	Surface tension	$N \cdot m^{-1}$
τ	Stress tensor (viscous or turbulent by context)	Pa
ε_D	Small regularisation parameter in drag relation	–

Subscripts

c	Continuous phase
d	Dispersed phase
l	Liquid
g	Gas (vapor)
w	Wall
sat	Saturation condition
k, ℓ	Generic phase indices
bub	Bubbly-regime quantity
drpl	Droplet-regime quantity
mix	Blended (mixture) quantity
sta	Stationary-bubble contribution
sli	Sliding bubble contribution
eva	Evaporative contribution
conv	Convective contribution
sub	Subcooled (e.g., Ja_{sub})
sup	Superheat (e.g., Ja_{sup})
int	Interface (e.g., ΔT_{int})

References

1. Reay, D.A.; Kew, P.A.; McGlen, R. *Heat Pipes: Theory, Design and Applications*, 6th ed.; Butterworth-Heinemann: Oxford, UK, 2013.
2. Grover, G.M. Evaporation-Condensation Heat Transfer Device. U.S. Patent 3229759A, 1966.
3. Collier, J.G.; Thome, J.R. *Convective Boiling and Condensation*, 3rd ed.; Clarendon Press: Oxford, UK, 1994.
4. Kharangate, C.R.; Mudawar, I. Review: Review of computational studies on boiling and condensation. *Int. J. Heat Mass Transf.* **2017**, *108*, 1164–1196. [\[CrossRef\]](#)
5. Bergman, T.L.; Lavine, A.; Incropera, F.P.; DeWitt, D.P. *Fundamentals of Heat and Mass Transfer*, 7th ed.; John Wiley & Sons: Hoboken, NJ, USA, 2011.
6. Fadhl, B. Modelling of the Thermal Behaviour of a Two-Phase Closed Thermosyphon. Ph.D. Thesis, Brunel University, Uxbridge, UK, 2016.
7. Petersen, K.J.; Rahbarimanesh, S.; Brinkerhoff, J.R. Progress in physical modelling and numerical simulation of phase transitions in cryogenic pool boiling and cavitation. *Appl. Math. Model.* **2023**, *116*, 327–349. [\[CrossRef\]](#)
8. Devahdhanush, V.S.; Lei, Y.; Chen, Z.; Mudawar, I. Assessing advantages and disadvantages of macro- and micro-channel flow boiling for high-heat-flux thermal management using computational and theoretical/empirical methods. *Int. J. Heat Mass Transf.* **2021**, *169*, 120787. [\[CrossRef\]](#)
9. Han, J.; He, W.; Liu, Y.; Gao, D.; Zhao, C.; Bo, H. Numerical investigation of flow boiling characteristics in narrow rectangular channels using two-fluid Eulerian CFD model covering a wide range of pressures. *Appl. Therm. Eng.* **2024**, *255*, 123982. [\[CrossRef\]](#)
10. Fadhl, B.; Wrobel, L.C.; Jouhara, H. Numerical modelling of the temperature distribution in a two-phase closed thermosyphon. *Appl. Therm. Eng.* **2013**, *60*, 122–131. [\[CrossRef\]](#)
11. Wang, Z.; Turan, A.; Craft, T. Numerical Investigation of the Two-Phase Closed Thermosyphon Operating with Non-Uniform Heat Flux Profiles. *Energies* **2023**, *16*, 5141. [\[CrossRef\]](#)
12. Edelbauer, W. Numerical simulation of cavitating injector flow and liquid spray break-up by combination of Eulerian–Eulerian and Volume-of-Fluid methods. *Comput. Fluids* **2017**, *144*, 19–33. [\[CrossRef\]](#)
13. Edelbauer, W.; Birkhold, F.; Rankel, T.; Pavlović, Z.; Kolar, P. Simulation of the liquid break-up at an AdBlue injector with the volume-of-fluid method followed by off-line coupled Lagrangian particle tracking. *Comput. Fluids* **2017**, *157*, 294–311. [\[CrossRef\]](#)
14. Kurul, N.; Podowski, M.Z. Multidimensional effects in force convection subcooled boiling. In Proceedings of the Ninth International Heat Transfer Conference, Jerusalem, Israel, 19–24 August 1990; pp. 21–26.
15. Sadeghi, K.; Kahani, M.; Ahmadi, M.H.; Zamen, M. CFD Modelling and Visual Analysis of Heat Transfer and Flow Pattern in a Vertical Two-Phase Closed Thermosyphon for Moderate-Temperature Application. *Energies* **2022**, *15*, 8955. [\[CrossRef\]](#)
16. Yang, D.; Bu, Z.; Jiao, B.; Wang, B.; Gan, Z. Numerical Study of Heat Transfer and Fluid Flow Characteristics of a Hydrogen Pulsating Heat Pipe with Medium Filling Ratio. *Energies* **2024**, *17*, 2697. [\[CrossRef\]](#)
17. Cheng, Y.; Yu, H.; Zhang, Y.; Zhang, S.; Shi, Z.; Xie, J.; Zhang, S.; Liu, C. Heat Transfer Mechanism Study of an Embedded Heat Pipe for New Energy Consumption System Enhancement. *Energies* **2024**, *17*, 6162. [\[CrossRef\]](#)
18. Godino, D.M.; Corzo, S.F.; Ramajo, D.E. CFD simulation of conjugated heat transfer with full boiling in OpenFOAM(R). *Appl. Therm. Eng.* **2022**, *213*, 118627. [\[CrossRef\]](#)

19. Gilman, L.A. Development of a General Purpose Subgrid Wall Boiling Model from Improved Physical Understanding for Use in Computational Fluid Dynamics. Ph.D. Thesis, Massachusetts Institute of Technology, Cambridge, MA, USA, 2014.
20. Shi, J.; Zhang, R.; Zhu, Z.; Ren, T.; Yan, C. A modified wall boiling model considering sliding bubbles based on the RPI wall boiling model. *Int. J. Heat Mass Transf.* **2020**, *154*, 119776. [[CrossRef](#)]
21. Jouhara, H.; Fadhl, B.; Wrobel, L.C. Three-dimensional CFD simulation of geyser boiling in a two-phase closed thermosyphon. *Int. J. Hydrogen Energy* **2016**, *41*, 16463–16476. [[CrossRef](#)]
22. Štručl, J.; Marn, J.; Zdravec, M. Numerical modelling of conjugate heat transfer in force convective flow environment using a combined boiling model (CBM). *Results Eng.* **2025**, *26*, 104958. [[CrossRef](#)]
23. Kopun, R.; Škerget, L.; Hriberšek, M.; Zhang, D.; Stauder, B.; Greif, D. Numerical simulation of immersion quenching process for cast aluminium part at different pool temperatures. *Appl. Therm. Eng.* **2014**, *65*, 74–84. [[CrossRef](#)]
24. Edelbauer, W.; Zhang, D.; Kopun, R.; Stauder, B. Numerical and experimental investigation of the spray quenching process with an Euler-Eulerian multi-fluid model. *Appl. Therm. Eng.* **2016**, *100*, 1259–1273. [[CrossRef](#)]
25. Srinivasan, V.; Kopun, R. Progress in the modeling and simulation of flow boiling heat transfer of binary fluids using advanced multi-fluid modeling techniques. In Proceedings of the ASME 2012 Fluids Engineering Division Summer Meeting, Rio Grande, PR, USA, 8–12 July 2012.
26. AVL List GmbH. *AVL FIRE v2018: User's Manual*; AVL List GmbH: Graz, Austria, 2018.
27. Greif, D.; Štručl, J. Numerical Study of Transient Multi Component Fuel Injection. In Proceedings of the SAE/KSAE 2013 International Powertrains, Fuels & Lubricants Meeting, Seoul, Republic of Korea, 21–23 October 2013; Volume 2013, p. 2550. [[CrossRef](#)]
28. Ishii, M. Thermo-Fluid Dynamic Theory of Two-Phase Flow. 1975. Available online: <https://ui.adsabs.harvard.edu/abs/1975STIA...7529657I/abstract> (accessed on 10 November 2025).
29. Lahey, R.T.; Drew, D.A. The analysis of two-phase flow and heat transfer using a multidimensional, four-field, two-fluid model. *Nucl. Eng. Des.* **2001**, *204*, 29–44. [[CrossRef](#)]
30. Drew, D.A.; Passman, S.L. *Theory of Multicomponent Fluids*; Springer: New York, NY, USA, 1998.
31. Ranz, W.E.; Marshall, W.R.J. Evaporation from Drops. *Chem. Eng. Prog.* **1952**, *48*, 141–146.
32. Mimouni, S.; Denèfle, R.; Gouénard-Fleau, S.; Vincent, S. Multifield Approach and Interface Locating Method for Two-Phase Flows in Nuclear Power Plant. In *Advances in Hydroinformatics*; Springer: Singapore, 2016; pp. 483–500.
33. Schiller, L.; Naumann, Z. Über die Grundlegenden Berechnungen bei der Schwerkraftaufbereitung. *Z. Ver. Dtsch. Ing.* **1933**, *77*, 318–320.
34. Tomiyama, A.; Kataoka, I.; Zun, I.; Sakaguchi, T. Drag Coefficients of Single Bubbles under Normal and Micro Gravity Conditions. *JSME Int. J. Ser. B* **1998**, *41*, 472. [[CrossRef](#)]
35. Robert, C. A photographic study of pool boiling in the region of the critical heat flux. *AIChE J.* **1960**, *6*, 533–538. [[CrossRef](#)]
36. Lemmert, M.; Chawla, L.M. Influence of flow velocity on surface boiling heat transfer coefficient. In *Heat Transfer Boiling*; Academic Press: Cambridge, MA, USA, 1977; pp. 237–247.
37. Tolubinsky, V.I.; Kostanchuk, D.M. Vapour bubbles growth rate and heat transfer intensity at subcooled water boiling. In Proceedings of the International Heat Transfer Conference 4, Paris-Versailles, France, 31 August–5 September 1970.
38. Basu, N.; Warriar, G.R.; Dhira, V.K. Wall Heat Flux Partitioning During Subcooled Flow Boiling: Part 1—Model Development. *J. Heat Transf.* **2005**, *127*, 131–140. [[CrossRef](#)]
39. Del Valle, V.H.; Kenning, D.B.R. Subcooled flow boiling at high heat flux. *Int. J. Heat Mass Transf.* **1985**, *28*, 1907–1920. [[CrossRef](#)]
40. Nusselt, W. Die Oberflächenkondensation des Wasserdampfes. *VDI* **1916**, *60*, 541–546.
41. Faghri, A. *Heat Pipe Science And Technology*; Taylor & Francis: Abingdon, UK, 1995.
42. Peterson, G.P. *An Introduction to Heat Pipes: Modeling, Testing, and Applications*; Wiley: Hoboken, NJ, USA, 1994.
43. Chen, Q.; Shi, Y.; Zhuang, Z.; Weng, L.; Xu, C.; Zhou, J. Numerical Analysis of Liquid–Liquid Heat Pipe Heat Exchanger Based on a Novel Model. *Energies* **2021**, *14*, 589. [[CrossRef](#)]
44. Badache, M.; Aidoun, Z.; Eslami-Nejad, P.; Blessent, D. Ground-Coupled Natural Circulating Devices (Thermosiphons): A Review of Modeling, Experimental and Development Studies. *Inventions* **2019**, *4*, 14. [[CrossRef](#)]
45. Lee, W.H. Pressure iteration scheme for two-phase flow modeling. *Multiph. Transp. Fundam. React. Saf. Appl.* **1980**, *1*, 407–431.
46. Wu, H.L.; Peng, X.F.; Ye, P.; Eric Gong, Y. Simulation of refrigerant flow boiling in serpentine tubes. *Int. J. Heat Mass Transf.* **2007**, *50*, 1186–1195. [[CrossRef](#)]
47. De Schepper, S.C.K.; Heynderickx, G.J.; Marin, G.B. Modeling the evaporation of a hydrocarbon feedstock in the convection section of a steam cracker. *Comput. Chem. Eng.* **2009**, *33*, 122–132. [[CrossRef](#)]
48. Vadlamudi, S.R.G.; Nayak, A.K. CFD simulation of Departure from Nucleate Boiling in vertical tubes under high pressure and high flow conditions. *Nucl. Eng. Des.* **2019**, *352*, 110150. [[CrossRef](#)]
49. Sweby, P.K. High Resolution Schemes Using Flux Limiters for Hyperbolic Conservation Laws. *SIAM J. Numer. Anal.* **1984**, *21*, 995–1011. [[CrossRef](#)]

50. Kocamustafaogullari, G. Pressure dependence of bubble departure diameter for water. *Int. Commun. Heat Mass Transf.* **1983**, *10*, 501–509. [[CrossRef](#)]
51. Wang, C.; Yao, F.; Shi, J.; Wu, L.; Zhang, M. Visualization Study on Thermo-Hydrodynamic Behaviors of a Flat Two-Phase Thermosyphon. *Energies* **2018**, *11*, 2295. [[CrossRef](#)]
52. Li, J.; Wang, R.; Xia, T.; Chen, H. Visualization Investigation of Heat Transfer Behavior in a Flat-Tube Shaped Heat Pipe. *Energies* **2025**, *18*, 1219. [[CrossRef](#)]
53. Courant, R.; Friedrichs, K.; Lewy, H. Über die partiellen Differenzengleichungen der mathematischen Physik. *Math. Ann.* **1928**, *100*, 32. [[CrossRef](#)]

Disclaimer/Publisher’s Note: The statements, opinions and data contained in all publications are solely those of the individual author(s) and contributor(s) and not of MDPI and/or the editor(s). MDPI and/or the editor(s) disclaim responsibility for any injury to people or property resulting from any ideas, methods, instructions or products referred to in the content.