

Research paper

An accurate low-order model of a fin-and-tube heat exchanger for the dynamic simulation of heat pumps

Roberto Sedoni^{a,*}, Paolo E. Santangelo^{a,b}, Diego Angeli^{a,b}

^a Dipartimento di Scienze e Metodi dell'Ingegneria, Università degli studi di Modena e Reggio Emilia, Via Amendola 2, 42122, Reggio Emilia, Italy

^b Centro Interdipartimentale En&Tech, Università degli studi di Modena e Reggio Emilia, Piazzale Europa 1, 42124, Reggio Emilia, Italy

ARTICLE INFO

Keywords:

Fin&tube heat exchanger
Segment-by-segment model
Low-order model
Dynamic model
Heat and mass transfer

ABSTRACT

Dynamic modeling of heat exchangers is essential for the analysis, control, and optimization of vapor compression systems such as heat pumps and air conditioners. These technologies are increasingly relevant in the context of energy transition, where regulatory frameworks and climate goals demand improved energy efficiency and reduced greenhouse gas emissions. Accurate yet computationally efficient models are crucial for system-level simulation, control design, and digital twin development. Traditional low-order approaches, such as lumped-parameter and moving boundary models, offer fast computation but often lack the fidelity needed to capture dynamic two-phase phenomena, especially under transient or off-design conditions. Finite volume models offer better accuracy but are often too computationally demanding for real-time applications. This work presents a segment-by-segment heat exchanger model that balances accuracy and efficiency, allowing for detailed prediction of dynamic behavior with limited computational cost. The model was developed using the Simscape language within the Simulink[®] environment and validated in both condenser and evaporator modes. Numerical comparisons with EVAP-COND 5.0 showed a maximum deviation of 2.7% for sensible heat transfer in condenser mode, and 6.8% and -2.4% for latent and sensible heat transfer in evaporator mode, respectively. Moreover, steady-state experimental validation corroborated these results, with overall heat flow rate predictions consistently within a $\pm 10\%$ error band. Thanks to this dual numerical and experimental validation, the proposed model provides a reliable tool for designing advanced, energy-efficient HVAC systems under stringent regulatory frameworks.

1. Introduction

Heat pumps and air conditioners constitute a major share of building energy consumption in both residential and commercial applications. As a result, improving their energy performance, operational flexibility, and reliability has been the subject of extensive research over the past decades. More recently, digital twins have gained increasing attention as a tool for the monitoring, analysis, and optimization of advanced HVAC systems. By combining physics-based or data-driven models with real-time operational data, digital twins can support performance assessment, fault detection and diagnosis, predictive maintenance, and operational optimization [1]. At the same time, advanced control strategies, particularly model predictive control (MPC), have attracted increasing interest for heat pump applications. By embedding dynamic system models directly into the control formulation, MPC can systematically optimize energy efficiency, operational flexibility, and occupant comfort [2–4]. In both cases, the quality of the results depends on the availability of dynamic models capable of accurately reproducing the transient behaviour of the system.

Within vapor-compression systems, heat exchangers play a central role in this regard. Indeed, they account for a substantial portion of the system thermal inertia and are responsible for much of its transient behavior. Accurate prediction of their dynamic response is therefore essential for reliable system-level simulation, control, and optimization. However, developing dynamic models that can achieve a suitable degree of accuracy at a low computational cost is inherently difficult due to the complex phase-change processes occurring within these components. During transient operation, the boundaries between the liquid, two-phase, and vapor regions shift continuously, and the thermophysical properties of the refrigerant undergo drastic, nonlinear variations. While detailed high-fidelity computational fluid dynamics (CFD) models can capture these spatial and temporal phenomena with higher accuracy, they typically demand prohibitive computational resources for real-time simulations. Consequently, the development of heat exchanger models that achieve an effective balance between physical fidelity and computational efficiency remains an active and important research area.

Several low-order modeling approaches were developed in past years, ranging from lumped-parameter [5] models to moving boundary

* Corresponding author.

E-mail addresses: roberto.sedoni@unimore.it (R. Sedoni), paoloemilio.santangelo@unimore.it (P.E. Santangelo), diego.angeli@unimore.it (D. Angeli).

Nomenclature			
<i>Latin letters</i>		ε	effectiveness [-]
A	cross-section area [m ²]	ε_r	relative roughness [-]
C	heat capacity rate [W K ⁻¹]	η	efficiency [-]
D_h	hydraulic diameter [m]	ν	specific volume [m ³ kg ⁻¹]
L	length [m]	ϕ	phase multiplier [-]
M	mass [kg]	Φ	total energy [W]
S	surface area [m ²]	ρ	density [kg m ⁻³]
T	temperature [K]	τ	time constant [s]
V	volume [m ³]	<i>Subscripts</i>	
W	humidity ratio [-]	$2P$	two-phase fluid
W_s	saturation humidity ratio [-]	A, B	control volume ports
c_{wall}	wall specific heat [J kg ⁻¹ K ⁻¹]	I	internal node
c_p	specific heat at constant pressure [J kg ⁻¹ K ⁻¹]	MA	moist air
f	Darcy friction factor [-]	M	mixture
$\tilde{f}$	Fanning friction factor [-]	S	single-phase
h	specific enthalpy [J kg ⁻¹]	SL	saturated liquid
k	thermal conductivity [W m ⁻¹ K ⁻¹]	SV	saturated vapor
$\dot{m}$	mass flow rate [kg s ⁻¹]	V	volume
Nu	Nusselt number [-]	avg	average
NTU	Number of Transfer Units [-]	c	condensation
p	pressure [Pa]	$cond$	condensed water
Pr	Prandtl number [-]	dry	dry air
$\dot{Q}$	heat flow rate [W]	eff	effective
Re	Reynolds number [-]	in	inlet
t	time [s]	out	outlet
u	specific internal energy [J kg ⁻¹]	s	sensible
x	vapor quality [-]	w	water
x_{ws}	specific saturation humidity [-]	wall	wall
<i>Greek letters</i>		<i>Superscripts</i>	
α	heat transfer coefficient [W m ⁻² K ⁻¹]	V	volume
Δh_v	latent heat of evaporation [J kg ⁻¹]	std	steady-state
		W	wall

[6,7] and finite volume formulations [8,9]. Lumped-parameter models offer high computational efficiency with minimal state variables, making them suitable for real-time simulations, especially when embedded in model-based control strategies or hardware-in-the-loop environments. Nonetheless, their strong assumptions, such as uniform temperature and pressure distributions within control volumes, can significantly limit their predictive capability in two-phase flow conditions, where rapid changes in temperature, pressure, and refrigerant phase strongly influence the performance of the heat exchanger. These limitations become particularly critical when such models are used to evaluate transient behaviors, capacity modulation, or startup/shutdown cycles, where phase-change dynamics are dominant.

For this reason, more detailed yet still computationally efficient low-order models, such as moving boundary or discretized finite volume models, are often preferred in applications where higher fidelity is required. Moving boundary models offer a compromise between lumped and fully distributed models, by subdividing the domain into three zones and tracking the interfaces between these regions (e.g., subcooled, two-phase, superheated), for a more realistic description of phase-change dynamics. However, their structure is tightly linked to predefined flow regimes, making them less robust under off-design or transient conditions [10,11]. In addition, since each region is modeled as a single control volume, the local wall temperature is typically approximated in a coarse way, which limits the accuracy in evaluating wall condensation phenomena. This limitation becomes even more critical under winter conditions, when the heat exchanger model is possibly coupled with a frosting model, as an accurate prediction of local surface temperatures is essential for reliably estimating frost formation and its impact on heat

exchanger performance [12,13]. Moreover, pressure drop predictions can also be inaccurate, as they are highly sensitive to the local thermodynamic state of the refrigerant, especially during phase transitions.

In contrast, finite volume models—also known as segment-by-segment models—discretize the entire domain into a fixed grid of segments, enabling the accurate reconstruction of the refrigerant thermodynamic state along the flow path. This spatial resolution allows for a precise evaluation of local heat transfer coefficients, pressure drops, and wall temperature distributions, which vary significantly across different phase regimes and are critical for predicting the heat exchanger performance. However, achieving sufficient accuracy often requires a high number of control volumes, resulting in increased computational cost [14]. This modeling approach constitutes the backbone of established commercial software such as CoilDesigner® [15] and EVAP-COND [16], which are extensively employed for detailed geometric design and steady-state performance rating of fin-and-tube heat exchangers. Yet, since these tools are inherently formulated for steady-state conditions, they are ill-suited for capturing transient behaviors. This fundamental limitation, compounded by their closed-source architecture and prohibitive computational cost, effectively precludes the direct integration of such models into dynamic system-level simulations or real-time control environments.

While segment-by-segment formulations are highly effective for conventional heat exchanger configurations, high-fidelity multi-dimensional CFD simulations often remain the only viable approach for unconventional geometries or extreme operating conditions. However, due to high computational cost, these simulations are typically restricted to isolated single modules or representative periodic sections rather than

full-scale components, resolving fluid flow and heat transfer phenomena at a strictly local scale. In recent years, this capability has been exploited across a wide range of applications, such as the thermal design of heat exchangers subjected to hypersonic aerodynamic heating [17], the geometric optimization of polymer-based tube bundles [18], and the analysis of dynamic boundary conditions in air-source heat-pump drying systems, where CFD revealed the competing influence of wind speed and humidity on system COP [19]. CFD has also been extensively employed to characterize enhanced non standard heat-transfer surfaces. For example, Mehrjardi et al. [20] used detailed three-dimensional simulations to quantify the heat-transfer and pressure-drop characteristics of dimpled tubes and subsequently derived compact P-NTU correlations, enabling rapid performance predictions without the need for repeated CFD analyses. This approach reflects a broader trend toward deriving reduced-order models from high-fidelity simulations, retaining much of their predictive capability while substantially reducing computational cost. The motivation is clear: despite their high predictive fidelity, CFD simulations often require several hours or even days for a single operating condition, making them impractical for transient system-level simulations, optimization studies, and real-time control applications.

To overcome this computational bottleneck while preserving predictive accuracy, a growing body of recent research has focused on data-driven and physics-informed surrogate models trained on high-fidelity datasets. Architectures such as physics-informed neural networks (PINNs) [21,22], hybrid convolutional-physics models [23], and deep learning frameworks specifically targeting heat exchangers in vapor compression cycles [24,25] have demonstrated computation times several orders of magnitude shorter than CFD while retaining high predictive accuracy. Despite these promising results, purely data-driven surrogates typically demand extensive and diverse training datasets, offer limited physical interpretability, and often show reduced predictive capability when applied beyond the range of operating conditions included in the training data. These limitations can compromise their reliability for model-based control, system optimization, and other applications requiring robust extrapolation to off-design conditions.

In light of the aforementioned limitations, the present study proposes a dynamic, segment-by-segment heat exchanger model designed to reconcile high-fidelity spatial discretization with the computational demands of real-time, system-level simulations. The framework is developed natively in the Simscape language within the Simulink® environment, adopting a flexible, white-box architecture that ensures complete transparency of the applied governing equations and empirical correlations. This formulation explicitly accounts for transient phenomena, including thermal inertia, refrigerant mass accumulation, and phase-change transitions, thereby enabling an accurate dynamic characterization of the component under time-varying operating conditions. To overcome the limitations of existing tools, the source code is provided in a public repository [26]. This open-source approach ensures reproducibility and allows researchers to easily adapt the model to their specific needs. Furthermore, the native Simscape architecture facilitates direct integration with standard libraries, enabling detailed heat exchanger models to be coupled directly with control algorithms without the need for complex co-simulation interfaces. While preserving high physical fidelity, the model is numerically optimized to reduce computational cost, making it highly effective for real-time applications, including system-level digital twins, model-based control design, and hardware-in-the-loop testing.

2. Model

2.1. Refrigerant finite volume

To develop a physically accurate yet computationally efficient model of a moist-air (MA) to two-phase flow (2P) heat exchanger, the refrigerant side is discretized into a sequence of finite control volumes (segments) along the main flow direction. Each segment is treated as a fixed

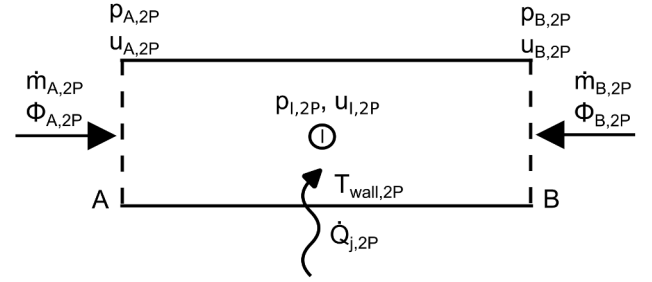


Fig. 1. Finite volume for the refrigerant stream.

volume (control volume) of fluid in which the mass, momentum, and energy balance equations are solved. To simplify the model and reduce the computational cost, a one-dimensional flow assumption is adopted along the axial direction; multidimensional and cross-flow effects are instead accounted for at the segment level through local effectiveness formulations and empirical friction factor and heat transfer correlations. In addition, the model explicitly accounts for the dynamic compressibility and thermal inertia of the refrigerant as well as the thermal inertia of the heat exchanger wall.

Fig. 1 illustrates the refrigerant finite volume, showing the positive direction of relevant variables: mass flow rates $\dot{m}$, total energy Φ (sum of enthalpy and kinetic energy), and the total heat flow rate that enters the segment $\dot{Q}_{j,2P}$. This generalized sign convention allows the model to inherently handle flow reversal without requiring substantial modifications to the governing equations.

An internal node, labeled as 'I', is defined within the segment, and the state variables describing the fluid's state (p and u) within the control volume are associated with this node.

Mass balance

In heat pump systems involving phase-changing refrigerants, local variations in pressure and enthalpy can also cause significant density changes, especially during phase transitions. For this reason, the dynamic compressibility of the refrigerant is taken into account, allowing the mass conservation equation to reflect transient mass accumulation within the segment. Consequently, the inlet and outlet mass flow rates may differ during dynamic conditions. The mass conservation equation for the refrigerant, under the one-dimensional flow assumption, in a segment, reads as:

$$\frac{dM_{2P}}{dt} = \dot{m}_{A,2P} + \dot{m}_{B,2P} \quad (1)$$

where $M_{2P} = \rho_{I,2P} V_{2P}$ is the refrigerant mass in the control volume. Then, applying the chain rule to express the time derivative of the fluid volume density ($\rho_{I,2P}$) as a function of pressure and specific internal energy, an expression for the rate of mass accumulation within the segment is obtained:

$$\frac{dM_{2P}}{dt} = \frac{d\rho_{I,2P}}{dt} V_{2P} = \left[\left(\frac{\partial \rho_{I,2P}}{\partial p_{I,2P}} \right)_{u_{I,2P}} \dot{p}_{I,2P} + \left(\frac{\partial \rho_{I,2P}}{\partial u_{I,2P}} \right)_{p_{I,2P}} \dot{u}_{I,2P} \right] V_{2P} \quad (2)$$

The partial derivatives of density with respect to pressure and specific internal energy in Eq. (2) are treated as state properties of the two-phase fluid domain and, as such, are calculated *a priori* and conveniently stored in a lookup table.

Energy balance The energy conservation equation for the refrigerant control volume is:

$$\frac{d(M_{2P} u_{I,2P})}{dt} = \Phi_{A,2P} + \Phi_{B,2P} + \dot{Q}_{j,2P} \quad (3)$$

Applying the product rule to the left-hand side of Eq. (3), the final energy balance equation of the segment becomes:

$$M_{2P} \frac{du_{I,2P}}{dt} + u_{I,2P}(\dot{m}_{A,2P} + \dot{m}_{B,2P}) = \Phi_{A,2P} + \Phi_{B,2P} + \dot{Q}_{j,2P} \quad (4)$$

Heat transfer from the wall to the fluid is expressed by Newton's law of cooling:

$$\dot{Q}_{j,2P} = \alpha_{2P} S_{2P} (T_{wall,2P} - T_{I,2P}) \quad (5)$$

where S_{2P} and α_{2P} are the segment surface area and the heat transfer coefficient at the refrigerant side.

Heat transfer coefficient

The refrigerant-side heat transfer coefficient is evaluated according to the local flow regime. In single-phase flow (subcooled liquid or superheated vapor), under turbulent conditions ($Re \geq 4000$), the Gnielinski [27] correlation is used:

$$\alpha = \frac{k_I}{D_h} \frac{\frac{f}{8} (Re_I - 1000) Pr_I}{1 + 12.7 \sqrt{\frac{f}{8}} (Pr_I^{2/3} - 1)} \quad (6)$$

In two-phase flow, under turbulent conditions, the Cavallini-Zecchin (if the refrigerant is condensing) [28] or Shah (if the refrigerant is evaporating) [29] correlations are used:

$$\alpha = \frac{k_{SL}}{D_h} \begin{cases} 0.05 \left[(1 - x_I + x_I \sqrt{v_{SV}/v_{SL}}) Re_{SL} \right]^{0.8} Pr_{SL}^{0.33} & \text{Cavallini-Zecchin} \\ 1.8 \frac{0.023 [Re_{SL}(1 - x_I)]^{0.8} Pr_{SL}^{0.4}}{[(1/x_I - 1)^{0.8} (v_{SL}/v_{SV})^{0.5}]^{0.8}} & \text{Shah} \end{cases} \quad (7)$$

where $Re_{SL} = \frac{\dot{m}_{avg} D_h}{A_{2P} \mu_{SL}}$.

Under laminar flow conditions ($Re < 2000$), the Nusselt number is approximated as a constant 3.66, which is the characteristic value for fully developed flow in circular tubes under a constant wall temperature boundary condition [30]. Within the transition region ($2000 \leq Re \leq 4000$), this constant value is blended via Eq. (8) with the turbulent Nusselt numbers evaluated using the correlations of Gnielinski [27], Cavallini-Zecchin [28], and Shah [29].

Although two-phase heat transfer correlations are generally applicable to single-phase conditions, they often yield significant errors, as shown by Santa [31]. For this reason, Eq. (6) is used when the fluid is entirely liquid or vapor. However, direct switching between single- and two-phase correlations produces discontinuities in the heat transfer coefficient (Fig. 2). To ensure smooth transitions at $x_I \rightarrow 0$ and $x_I \rightarrow 1$,

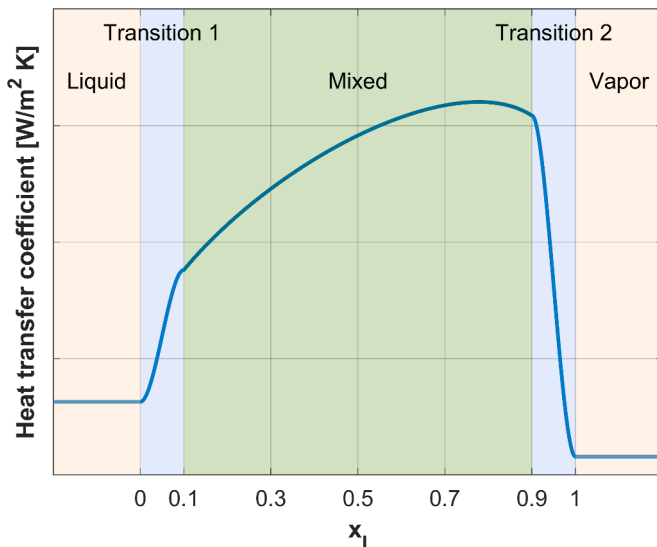


Fig. 2. Blending of the heat transfer coefficient across flow regimes.

a cubic Hermite spline is applied over narrow vapor quality intervals $x_I \in \{[0, 0.1], [0.9, 1]\}$:

$$\alpha = (1 - 3\theta^2 + 2\theta^3) \alpha(x_1) + (3\theta^2 - 2\theta^3) \alpha(x_2), \quad \theta = \frac{x_I - x_1}{x_2 - x_1} \quad (8)$$

Here x_1 and x_2 denote the endpoints of each transition interval; $\alpha(x_1)$ and $\alpha(x_2)$ are evaluated from the appropriate correlation (single- or two-phase). Outside the transition region, the base correlation is used.

While conventional treatments typically rely on complex flow-regime tracking to handle phase changes, the proposed strategy bypasses these computationally expensive methods. Instead, it introduces a straightforward blending approach restricted strictly to predefined boundary zones. This targeted smoothing effectively eliminates artificial mathematical discontinuities and provides a continuous, accurate representation of the transitional segments (i.e., segments where phase change occurs). By drastically reducing algorithmic complexity, this implementation achieves robust numerical stability and exceptional computational efficiency, thereby supporting rapid transient execution times, as shown in the subsequent sections, without sacrificing overall accuracy.

2.1.1. Mechanical energy balance

Since the inlet and outlet mass flow rates may differ during dynamic conditions due to mass accumulation, the segment itself is subdivided into two parts, and the pressure losses are calculated separately for each section, assuming the flow rates in the two sections are $\dot{m}_{A,2P}$ and $\dot{m}_{B,2P}$, respectively. For the half adjacent to the port side (A or B), the mechanical energy balance equation is:

$$p_{i,2P} - p_{I,2P} = \frac{\dot{m}_{i,2P}}{A_{2P}} \left| \frac{\dot{m}_{i,2P}}{A_{2P}} (v_{I,2P} - v_{i,2P}) \right| + \Delta p_{visc,i,2P} \quad (9)$$

The first term on the right-hand side represents momentum flux losses from mass entering at port i and the second term represents the viscous friction losses (Δp_{visc}) within the first or second half of the segment. As with the heat transfer coefficient, viscous friction losses may vary significantly depending on the fluid state [32]. Also in this case, the discretization approach allows for a more precise estimation of the total pressure drop.

In the context of phase-change flows, two distinct modeling approaches are commonly found in the literature: the homogeneous flow model and the separated flow model. The homogeneous flow model assumes that the liquid and vapor phases are perfectly mixed and move at the same velocity, effectively treating the two-phase mixture as a single pseudo-fluid with average properties. In contrast, the separated flow model treats the liquid and vapor phases as distinct, each with its own velocity and flow characteristics, requiring more complex modeling techniques. In case of the homogeneous model, viscous friction losses can be readily evaluated using the classical Darcy-Weisbach equation as follows:

$$\Delta p_{visc,i,2P} = \frac{\dot{m}_{i,2P} |\dot{m}_{i,2P}| f_{i,2P} L_{eff} v_{I,2P}}{4 D_{h,2P} A_{2P}} \quad (10)$$

Here, L_{eff} is the effective length of the segment (sum of the geometric length plus an aggregate equivalent length that accounts for the local resistance of the segment), and $D_{h,2P}$ is the hydraulic diameter. The Darcy friction factors $f_{i,2P}$ can be calculated, for example, using the Haaland correlation [33]:

$$f_{i,2P} = \left\{ -1.8 \log_{10} \left[\left(\frac{6.9}{Re_{i,2P}} \right) + \left(\frac{\epsilon_r}{3.7} \right)^{1.11} \right] \right\}^{-2} \quad (11)$$

where ϵ_r is the relative roughness of the pipe and $Re_{i,2P} = \frac{\dot{m}_{i,2P} D_h}{A_{2P} \mu_{i,2P}}$ is the Reynolds number of the equivalent single-phase flow at the respective port.

In contrast, under the separated flow model, viscous pressure losses are often evaluated using the Lockhart-Martinelli correlation [34]:

$$\Delta p_{visc,i,2P} = \phi_{SL}^2 \Delta p_{SL} \quad (12)$$

where ϕ_{SL}^2 is the two-phase frictional multiplier for the liquid phase, defined as:

$$\phi_{SL}^2 = 1 + \frac{K}{X} + \frac{1}{X^2} \quad (13)$$

where $X^2 = \frac{\Delta p_{SL}}{\Delta p_{SV}}$ is the ratio between single-phase liquid (Δp_{SL}) and vapor (Δp_{SV}) pressure drops, better known as the Lockhart-Martinelli parameter. K is the Chisholm parameter, a factor used to account for the interfacial pressure drop between the liquid and gas phases, determined based on the flow regimes of the liquid and gas phases:

$$K = \begin{cases} 5 & \text{for } \text{Re}_{SL}^* < 1500 \text{ and } \text{Re}_{SV}^* < 1500 \\ 10 & \text{for } \text{Re}_{SL}^* \geq 1500 \text{ and } \text{Re}_{SV}^* < 1500 \\ 12 & \text{for } \text{Re}_{SL}^* < 1500 \text{ and } \text{Re}_{SV}^* \geq 1500 \\ 20 & \text{for } \text{Re}_{SL}^* \geq 1500 \text{ and } \text{Re}_{SV}^* \geq 1500 \end{cases} \quad (14)$$

The pressure drops Δp_k of each individual phase are calculated, again, using the Darcy-Weisbach equation:

$$\Delta p_k = \frac{\dot{m}_{i,k} |\dot{m}_{i,k}| f_{i,k} L_{\text{eff}} v_{I,k}}{4 D_h A_{2P}} \quad (15)$$

Here, $\dot{m}_{i,k}$ is the mass flow rate of the k th phase (i.e., liquid or vapor) entering the i th port. For instance, in the case of the vapor phase, it is expressed as: $\dot{m}_{i,SV} = \dot{m}_i x_i$ where x_i is the inlet vapor quality at port

i . Furthermore, $f_{i,k} = \left\{ -1.8 \log_{10} \left[\left(\frac{6.9}{\text{Re}_{i,k}^*} \right) + \left(\frac{\epsilon_r}{3.7} \right)^{1.11} \right] \right\}^{-2}$ is the friction factor of the k th phase with the phase Reynolds number defined as $\text{Re}_{i,k}^* = \frac{|\dot{m}_{i,k}| D_h}{A_{2P} \mu_{i,k}}$, and $v_{I,k}$, $\mu_{i,k}$ are the specific volume and dynamic viscosity of the k th phase at saturation, respectively, evaluated at the fluid volume pressure $p_{I,2P}$. Similar to the approach used for the heat transfer coefficient in the previous section, the separated flow model, specifically the Lockhart-Martinelli correlation, remains applicable across the full vapor quality range. However, as observed in [32], a simpler homogeneous flow model may yield more accurate results at the vapor quality extremes. This is because the assumption of a well-mixed flow becomes increasingly valid as the flow approaches these limits. To improve accuracy, the separated flow model is therefore blended with the homogeneous model in the transition regions (vapor quality between 0.0.1 and 0.9-1), following Eq. (8), as was done for the heat transfer coefficient. If the fluid volume is in a subcooled liquid or superheated vapor state (i.e., $x_i = 0$ or $x_i = 1$), viscous pressure losses are instead evaluated using the homogeneous model, Eq. (10), which corresponds to the conventional approach for modeling viscous friction losses in single-phase flows.

2.1.2. Moist-air side

On the moist-air side, a control volume analogous to the one on the refrigerant side is defined. The same positive directions for mass and energy flow rates shown in Fig. 1 are adopted. The fluid state within this volume is represented by an internal node 'I', defined by pressure (p), temperature (T), and specific humidity (x_w).

Mass balance

The mass conservation equations for each segment on the moist-air side are given by:

$$\text{Total mass: } \dot{m}_{A,MA} + \dot{m}_{B,MA} - (\dot{m}_{\text{cond}}^V + \dot{m}_{\text{cond}}^W) = 0 \quad (16)$$

$$\text{Water mass: } \dot{m}_{w,A,MA} + \dot{m}_{w,B,MA} - (\dot{m}_{\text{cond}}^V + \dot{m}_{\text{cond}}^W) = 0 \quad (17)$$

In these equations, the mass flow rates of condensed water in the bulk-flow ($\dot{m}_{\text{cond}}^V$) and at the wall surface ($\dot{m}_{\text{cond}}^W$) are considered positive when leaving the control volume. Given that the Mach number in heat pump applications remains below 0.3, air and water vapor compressibility effects are neglected in this case.

To avoid unphysical conditions during the cooling of the moist-air volume in the simulation, such as relative humidity exceeding 100%, the excess water vapor is assumed to condense in the bulk-flow and is then removed in liquid form from the control volume:

$$\dot{m}_{\text{cond}} = \begin{cases} 0 & x_{w,I} \leq x_{ws,I} \\ \frac{x_{w,I} - x_{ws,I}}{\tau_{\text{cond}}} \rho_I V_{MA} & x_{w,I} > x_{ws,I} \end{cases} \quad (18)$$

where τ_{cond} is the characteristic time constant of the condensation process, and $x_{ws,I}$ is the saturation humidity ratio at the control volume temperature $T_{I,MA}$. The energy term associated with this condensed water is:

$$\Phi_{\text{cond}} = \dot{m}_{\text{cond}}^V h_{\text{wl,cond}}^V \quad (19)$$

Here, $h_{\text{wl,cond}}^V$ is the specific enthalpy of the condensed water in the bulk-flow, evaluated at the control volume temperature $T_{I,MA}$.

To evaluate the influence of the condensation time constant (τ_{cond}) on the model transient predictions, a sensitivity analysis was conducted. The simulated case study employs the converged evaporator model (detailed in Section 3.1), subjected to an inlet air temperature step change. Specifically, the air-side boundary conditions transition from an initial state of 25°C and 80% relative humidity to a final state of 30°C and 80% relative humidity, maintaining a constant volumetric flow rate of 7.5 L/min. Concurrently, the refrigerant side is maintained at a constant saturation temperature of 5°C, with an inlet vapor quality of 0.2 and a mass flow rate of 40 kg/h. These operating conditions were specifically chosen to induce bulk-flow saturation and actively trigger condensation. Fig. 3 illustrates the impact of varying τ_{cond} from 10^{-5} s to 10^{-1} s on the total heat flow rate and outlet relative humidity.

The results indicate that under such highly condensing conditions, the choice of τ_{cond} influences the physical accuracy of the model during rapid transient cooling phases. As shown in Fig. 3b, excessively large values (e.g., 10^{-1} s or 10^{-2} s) fail to promptly remove the condensed water vapor. This temporal delay leads to unphysical thermodynamic states where the outlet relative humidity significantly exceeds the 100% saturation limit. Furthermore, this artificial lag in the condensation process alters the instantaneous latent heat release, causing noticeable deviations in the predicted total heat flow rate, as observed in Fig. 3a. Table 1

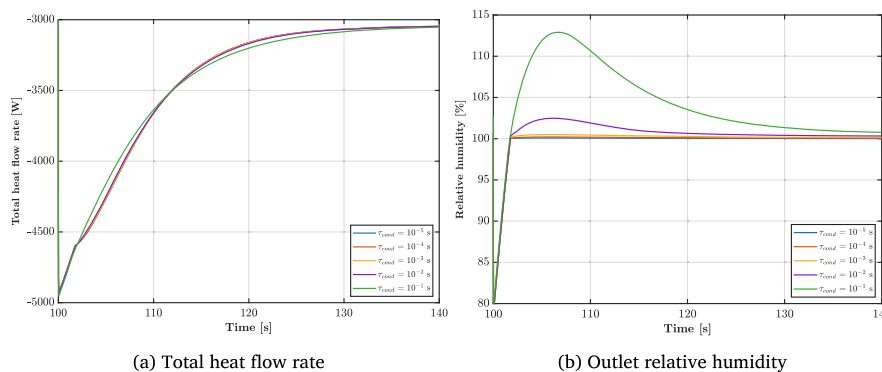


Fig. 3. Impact of the τ_{cond} parameter on the total heat flow rate and the outlet relative humidity under transient conditions.

Table 1

Maximum differences for $\dot{Q}_t$ and outlet RH compared to the reference $\tau_{\text{cond}} = 10^{-5}$.

τ_{cond}	Max Difference $\dot{Q}_t$	Max Difference RH
10^{-4}	0.7	0.12
10^{-3}	2.6	0.42
10^{-2}	20.0	2.39
10^{-1}	103.4	12.82

quantifies these discrepancies by comparing the maximum deviations against a reference value obtained for $\tau_{\text{cond}} = 10^{-5}$ s. The results demonstrate that for $\tau_{\text{cond}} \geq 10^{-2}$ s, the deviations become substantial, reaching up to 103.4 W for the total heat flow rate and 12.82% for the outlet relative humidity, for the case analyzed. Conversely, for $\tau_{\text{cond}} \leq 10^{-3}$ s, the model rapidly restores thermodynamic equilibrium, effectively maintaining the relative humidity near the saturation limit without introducing significant errors in the overall heat transfer rates.

Furthermore, it is worth noting that under steady-state conditions, the discrepancies in the total heat flow rate converge to zero across all tested configurations. However, a small residual error in the outlet relative humidity persists at steady state when using larger time constants. Specifically, the highest τ_{cond} value tested (10^{-1} s) yields a slight supersaturation error of +0.55%, an inaccuracy that tends to zero for $\tau_{\text{cond}} \leq 10^{-3}$ s.

Therefore, to ensure physical consistency during transient simulations while avoiding the numerical stiffness typically associated with infinitesimally small time constants, a value of $\tau_{\text{cond}} \leq 10^{-3}$ s is recommended. In the specific model presented, τ_{cond} is taken equal to 10^{-4} s.

Wall surface condensation

Condensation of water vapor from moist air on heat exchanger surfaces occurs when the wall surface temperature drops below the dew point.

The rate of water vapor condensing on the wall surface for each segment is evaluated under the assumption of uniform wall temperature, following [13]:

$$\dot{m}_{\text{cond}}^W = |\dot{m}_{\text{dry,MA}}| (W_{\text{in,MA}} - W_{\text{wall,MA}}) \left\{ 1 - \exp \left[- \frac{\alpha_{m,MA} (A_{\text{tube,MA}} + \eta_{f,\text{in,MA}} A_{f,\text{in}})}{|\dot{m}_{MA}|} \right] \right\} \quad (20)$$

where $\dot{m}_{\text{dry,MA}}$ is the dry-air mass flow rate. $W_{\text{in,MA}}$ is the inlet humidity ratio at the moist-air side, and $W_{\text{wall,MA}}$ is the humidity ratio at the wall, defined as:

$$W_{\text{wall,MA}} = \min(W_{\text{in,MA}}, W_{\text{sat,MA}}^W) \quad (21)$$

Here, $W_{\text{sat,MA}}^W$ is the saturation humidity ratio evaluated at the wall temperature on the air side of the segment. The min function in Eq. (21) provides the switch between “dry” and “wet” heat transfer:

- when the surface temperature is above the dew point, $W_{\text{sat,MA}}^W > W_{\text{in,MA}}$; therefore, condensation is not occurring ($\dot{m}_{\text{cond}}^W = 0$);
- when the surface temperature is below the dew point, $W_{\text{sat,MA}}^W < W_{\text{in,MA}}$; therefore, condensation is occurring ($\dot{m}_{\text{cond}}^W \neq 0$).

This condensate is assumed to be removed from the segment control volume. The energy term associated with the removal of this condensate is calculated as:

$$\Phi_{\text{cond}}^W = \dot{m}_{\text{cond}}^W h_{\text{wl,cond}} \quad (22)$$

where $h_{\text{wl,cond}}$ is the specific enthalpy of the removed liquid water evaluated at the average local wall temperature ($T_{\text{wall,MA}}$).

Energy balance

The energy conservation equation for the moist-air control volume is:

$$\Phi_{A,MA} + \Phi_{B,MA} + \dot{Q}_{j,MA} - (\Phi_{\text{cond}}^V + \Phi_{\text{cond}}^W) = 0 \quad (23)$$

where $\dot{Q}_{j,MA}$ is the segment heat flow rate transferred to the moist-air control volume. In Eq. (23), the thermal capacity of the moist air is

neglected, resulting in a quasi-steady-state formulation. This assumption is justified by the low density of the air stream and its brief residence time within the heat exchanger (typical face velocities are 1.5 to 3 m s⁻¹ in HVAC applications), which make its thermal inertia negligible compared to the thermal capacity of the solid structure.

Mechanical energy balance

The air side pressure loss across each segment ($\Delta p_{j,MA}$) is calculated using the Kays and London equation [35]:

$$\Delta p_{j,MA} = \frac{G_{MA}^2}{2 \rho_{\text{in,MA}}} \left[\tilde{f}_{MA} \frac{A_0}{A_c} \frac{\rho_{\text{in,MA}}}{\rho_{\text{avg,MA}}} + (1 + \sigma^2) \left(\frac{\rho_{\text{in,MA}}}{\rho_{\text{out,MA}}} - 1 \right) \right] \quad (24)$$

where G_{MA} is the moist-air mass flux per segment ($G_{MA} = \dot{m}_{\text{in,MA}}/A_c$), $\rho_{\text{in,MA}}$ is the segment inlet air density, $\rho_{\text{out,MA}}$ is the segment outlet air density, $\rho_{\text{avg,MA}}$ is their arithmetic mean, σ is the segment ratio of the minimum free flow area (A_c) to the segment heat exchange frontal-area and A_0 is the total segment surface area on the air side. The segment moist-air Fanning friction factor ($\tilde{f}_{MA}$) is obtained from the correlations of Wang for plain [36] or louvered fins [37]. For brevity, only the correlation for plain finned tubes is reported here:

$$\tilde{f}_{MA} = 0.0267 \text{Re}_{D_{c,MA}}^{F1} \left(\frac{P_t}{P_l} \right)^{F2} \left(\frac{F_p}{D_c} \right)^{F3} \quad (25)$$

where P_t is the transverse tube pitch, P_l is the longitudinal tube pitch, D_c is the tube outer diameter incremented by the fin thickness, F_p is the fin spacing, $\text{Re}_{D_{c,MA}}$ is the Reynolds number based on D_c and $F1, F2$ and $F3$ are correlation parameters defined in the reference article [36].

In order to maintain consistency with the employed empirical correlations, the Fanning friction factor is determined by considering the entire heat exchanger geometry and is then evaluated for each segment based on local flow conditions. Consequently, this method allows the one-dimensional formulation to intrinsically account for the complex multi-dimensional flow effects within each individual segment.

Heat and mass transfer coefficients

For each segment the Nusselt number on the moist-air side is calculated using the standard Colburn equation:

$$\text{Nu}_{MA} = j_{MA} \text{Re}_{D_{h,MA}} \text{Pr}_{MA}^{1/3} \quad (26)$$

with the Reynolds number ($\text{Re}_{D_{h,MA}}$) based on the hydraulic diameter (D_h) of the minimum flow area of the segment. The Colburn j -factor (j_{MA}) is obtained from the correlations of Wang for plain fins [36]:

for $N_r = 1$

$$j_{MA} = 0.108 \text{Re}_{D_{c,MA}}^{-0.29} \left(\frac{P_t}{P_l} \right)^{P1} \left(\frac{F_p}{D_c} \right)^{-1.084} \left(\frac{F_p}{D_h} \right)^{-0.786} \left(\frac{F_p}{P_t} \right)^{P2} \quad (27)$$

for $N_r \geq 1$

$$j_{MA} = 0.086 \text{Re}_{D_{c,MA}}^{P3} N_r^{P4} \left(\frac{F_p}{D_c} \right)^{P5} \left(\frac{F_p}{D_h} \right)^{P6} \left(\frac{F_p}{P_t} \right)^{-0.93} \quad (28)$$

where N_r is the number of tube rows, and $P1, \dots, P6$ are correlation parameters defined in the reference article [36]. Similar correlations are also implemented to determine the j -factor for the case of louvered fins [37].

The segment heat transfer coefficient on the air side can now be easily determined from the definition of the Nusselt number:

$$\alpha_{MA} = \frac{\text{Nu}_{MA} k_{MA}}{D_h} \quad (29)$$

Assuming the validity of the heat and mass transfer analogy, an estimation of the segment mass transfer coefficient is given by:

$$\alpha_{m,MA} = \frac{\alpha_{MA}}{c_{p,MA} \rho_{MA} \text{Le}^{2/3}} \quad (30)$$

where $\text{Le} = 0.9$ [38].

Similarly to the Fanning friction factor, to maintain consistency with the employed empirical correlations, j_{MA} is determined by considering

the entire heat exchanger geometry and is then evaluated for each segment based on local flow conditions, thereby intrinsically capturing the macroscopic 2D and 3D effects.

2.1.3. Overall segment heat transfer

At steady-state, the sensible heat flow rate ($\dot{Q}_{j,s}$) from the MA to the 2P side is computed for each segment using the effectiveness-NTU method:

$$\dot{Q}_{j,s} = \varepsilon C_{\min} (T_{in,MA} - T_{in,2P}) \quad (31)$$

where $C_{\min} = (\dot{m}c_p)_{\min}$ is the segment minimum heat capacity rate, $T_{in,MA}$ is the moist-air inlet temperature and $T_{in,2P}$ is the segment two-phase fluid inlet temperature. The effectiveness (ε) is calculated according to the flow arrangement. In the case of a fin-and-tube heat exchangers, the flow arrangement is typically *cross-flow* with both fluids unmixed:

$$\varepsilon = \begin{cases} 1 - \frac{\exp(C_r NTU^{0.78} - 1)}{C_r^{0.22} \exp(-NTU)} & \text{if single-phase} \\ 1 - \exp(-NTU) & \text{if two-phase} \end{cases} \quad (32)$$

with $C_r = \frac{C_{\min}}{C_{\max}}$ where $C_{\max}$ is the maximum heat capacity rate and $NTU = \frac{1}{R_{tot} C_{\min}}$. The same blending approach based on the vapor quality of the fluid volume (x_I), previously introduced for heat transfer and momentum balance on the refrigerant side, is also applied here to ensure a smooth transition between single-phase and two-phase effectiveness. This treatment enhances the numerical robustness of the model and improves simulation accuracy in transitional segments (i.e., segments where $0 \leq x_I \leq 0.1$ or $0.9 \leq x_I \leq 1$).

The total thermal resistance between the two flows (R_{tot}) is:

$$R_{tot} = R_{conv,2P} + R_{wall} + R_{conv,MA} \quad (33)$$

Thermal resistances are defined as: $R_{conv,2P} = \frac{1}{\alpha_{2P} A_{tube,2P}}$, $R_{wall} = \frac{\ln(\frac{OD}{ID})}{2\pi k_{tube} L_{tube}}$, $R_{conv,MA} = \frac{1}{\alpha_{MA} (A_{tube,MA} + \eta_{fin} A_{fin,MA})}$ for internal convective resistance ($R_{conv,2P}$), wall resistance (R_{wall}) and external convective resistance ($R_{conv,MA}$), respectively.

The fin efficiency (η_{fin}) is computed in this work using the approximated formula by Schmidt [39] for annular circular fins:

$$\eta_{fin} = \frac{\tanh\left[m\left(R_e - \frac{OD}{2}\right)\right]}{m\left(R_e - \frac{OD}{2}\right)} \quad (34)$$

where R_e is the equivalent fin external radius that can be determined using a 1D-SERF model [40], $m = \sqrt{\frac{\alpha_{MA}}{0.5 k_{fin} \delta_{fin}}}$ with k_{fin} the fin thermal conductivity and δ_{fin} the fin thickness.

Finally, the total heat flow rate transferred from moist air to the refrigerant, at steady-state, is:

$$\dot{Q}_j^{std} = \dot{Q}_{j,MA} = \dot{Q}_{j,2P} = \dot{Q}_{j,s} + \dot{Q}_{j,l} \quad (35)$$

where $\dot{Q}_{j,l} = \dot{m}_{cond}^W \Delta h_{vap}$ is the latent heat flow rate due to condensation that occurs on the moist-air side, and Δh_{vap} is the latent heat of vaporization of water evaluated at the wall temperature ($T_{wall,MA}$) of the segment. Note that in this modelling framework, sensible and latent heat are strictly interdependent, and their calculation requires iteration.

To simulate the dynamic behavior of the heat exchanger resulting from energy accumulation in the solid wall, a two-node thermal model is used. The wall is conceptually divided into two thermal masses, each representing the portion of wall material adjacent to moist-air (MA) and refrigerant (2P) control volumes. Although the physical thermal mass distribution in finned-tube heat exchangers is inherently asymmetric due to the external fins, an equal mass partitioning is adopted here as a lumped-parameter simplification. To rigorously preserve the total thermal capacitance of the component, the equivalent specific heat capacity

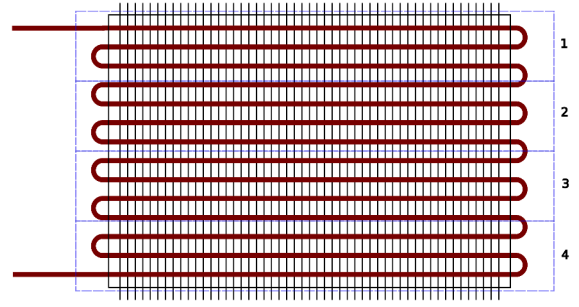


Fig. 4. Reconstruction of the single-row heat exchanger.

($\bar{c}_{wall}$) is computed as the mass-weighted average specific heat capacity of the tube and fin materials. Because this model is developed for system-level simulations, this formulation efficiently captures the macroscopic dynamic response and overall energy balances without introducing the computational overhead associated with asymmetric nodes.

The energy balances for the wall nodes read as follows:

$$\frac{M_{wall} \bar{c}_{wall}}{2} \frac{dT_{wall,2P}}{dt} = -\dot{Q}_{j,2P} - \dot{Q}_j^{std} \quad (36)$$

$$\frac{M_{wall} \bar{c}_{wall}}{2} \frac{dT_{wall,MA}}{dt} = -\dot{Q}_{j,MA} + \dot{Q}_j^{std} \quad (37)$$

where M_{wall} is the total wall mass, $\bar{c}_{wall}$ is the average specific heat capacity of the wall material, and $T_{wall,2P}$ and $T_{wall,MA}$ are the temperatures of the wall portions adjacent to two-phase and moist-air finite volumes, respectively.

The terms $\dot{Q}_{j,2P}$ and $\dot{Q}_{j,MA}$ represent the convective heat exchange between the wall surface and the two-phase and moist-air stream in each finite volume:

$$\dot{Q}_{j,2P} = \alpha_{2P} A_{tube,2P} (T_{wall,2P} - T_{I,2P}) \quad (38)$$

$$\dot{Q}_{j,MA} = \alpha_{MA} (A_{tube,MA} + \eta_{fin} A_{fin,MA}) (T_{wall,MA} - T_{I,MA}) \quad (39)$$

The sign of $\dot{Q}_j^{std}$ is taken as positive when heat flows from the wall node adjacent to “MA” to the wall node adjacent to “2P”.

2.1.4. Overall heat transfer

The individual heat exchanger segments can now be assembled to reconstruct the complete device under analysis. The core model developed in this work is a single-row heat exchanger unit, which functions as the primary building block for full multi-row devices. The following sections outline this assembly process, beginning with the single-row configuration and progressing to multi-row architectures.

As shown in Fig. 4, the geometry of the single-row heat exchanger is simply built by connecting the refrigerant control volumes of the segments in series, while the corresponding moist-air control volumes are arranged in parallel.

As a result, the total heat flow rate exchanged ($\dot{Q}_{tot,2P}$) and the overall pressure drop ($\Delta p_{tot,2P}$) on the refrigerant side for this configuration are given by the sum of the individual heat flow rates $\dot{Q}_{j,2P}$ and pressure drops $\Delta p_{j,2P}$ of the various segments.

On the other hand, for the moist-air side, the total heat flow rate exchanged ($\dot{Q}_{tot,MA}$) is likewise given by the sum of the individual heat flow rates ($\dot{Q}_{j,MA}$), while the overall pressure drop is approximated as:

$$\Delta p_{tot,MA} = \frac{\sum_{j=1}^{n_{seg,row}} \Delta p_{j,MA}}{n_{seg,row}} \quad (40)$$

where $n_{seg,row}$ is the number of segments per row. Eq. (40) is an approximation, as variations in air density caused by different heat flow rates across the segments would, according to Eq. (24), lead to different pressure drops on the air side of each segment. Since these segments are arranged in parallel on the air side, this would cause a redistribution of the airflow across the heat exchanger face to equalize the pressure

drops [12,41]. However, because density variations on the air side are relatively small, the resulting redistribution effect is limited and is neglected in this work. To complete the heat exchanger reconstruction, the previously built single-row units can be connected in series on both the air and refrigerant sides to form the multi-row, multi-pass configuration. Alternatively, for a multi-row single-pass configuration, the single-row units can be connected in parallel on the refrigerant side and in series on the air side.

In the context of this work, to simplify the model, adiabatic mixing is applied at the end of each row to compute an aggregate air outlet state of the single-row unit, which then serves as the inlet condition for the subsequent row. This approach allows different discretization levels to be adopted across the rows, enabling finer discretization only where needed while still capturing the temperature and humidity gradients along the moist-air side of the heat exchanger.

Modelling considerations for multi-circuit heat exchangers

Fin-and-tube heat exchangers are often designed with multiple circuits to improve thermal performance and reduce pressure drop. However, splitting the refrigerant flow into parallel paths can lead to maldistribution of the refrigerant, affecting the overall effectiveness. This issue is often caused by non-uniform airflow distribution across the heat exchanger, which leads to uneven thermal loads among the circuits. To mitigate this, flow distributors, optimized header geometries, and balancing devices such as orifices or capillary tubes are commonly used by manufacturers. To keep the computational cost of the model low, uniform airflow at the heat exchanger frontal face and symmetry among the refrigerant paths in multi-circuit heat exchangers can be assumed. This allows for a highly simplified modeling approach for this type of geometry: it is sufficient to evaluate heat transfer performance and pressure drops using the air and refrigerant mass flow rates divided by the number of parallel circuits (multi-circuit approximation). In the global energy balances of the heat exchanger model, the calculated exchanged heat for the single heat exchanger must then be multiplied by the number of parallel circuits. On the other hand, for pressure drops, the values calculated for a single parallel circuit are used directly in the global balances, since the composite circuit behaves as hydraulic resistances in parallel on both the air and refrigerant sides. If a balancing device is used, it is sufficient to adjust L_{eff} , on the refrigerant side, to account for the associated local resistance.

3. Results and discussion

3.1. Numerical comparison

A first validation of the fin-and-tube heat exchanger was performed via simulation. A heat exchanger with 3 tube rows and 15 tubes per row, composed of two parallel refrigerant circuits, was simulated. R32 was selected as the working fluid. The results of the developed model were compared against those obtained using EVAP-COND 5.0, a widely used steady-state simulation tool for air-to-refrigerant heat exchangers. EVAP-COND, developed by NIST, is based on a detailed segment-by-segment modeling of the heat exchanger geometry, accounting for refrigerant flow distribution, phase-change phenomena, and heat and mass transfer processes on both the air and refrigerant sides. To support the validity of the multi-circuit approximation, a uniform air velocity distribution was assumed across the frontal surface of the heat exchanger. Two different operating conditions for the heat exchanger were tested:

- **Condenser operating condition:** on the refrigerant side, an inlet mass flow rate of 40 kg h^{-1} , an inlet saturation temperature of 45°C , and a superheating of 30°C . On the air side, a volumetric flow rate of $450 \text{ m}^3\text{h}^{-1}$ (based on a standard air density of 1.204 kg m^{-3} , as defined in the EVAP-COND documentation [16]), with inlet air conditions of 22°C and 50 % relative humidity.

- **Evaporator operating condition:** on the refrigerant side, an inlet mass flow rate of 40 kg h^{-1} , an inlet saturation temperature of 7°C , and a vapor quality of 0.2. On the air side, a volumetric flow rate of $450 \text{ m}^3\text{h}^{-1}$, with inlet air conditions of 26°C and 50 % relative humidity.

Since the developed model employs a segment-by-segment approach, the comparison was performed by varying the number of segments. The coarsest discretization corresponded to the minimum resolution required to capture the temperature gradient on the moist-air (MA) side, using only one segment per row. The number of segments was then increased by one per row in sequence, starting from the air-inlet row. Conversely, within the EVAP-COND software, the discretization method was set to the finest resolution, corresponding to 10 segments per tube. To quantitatively evaluate the accuracy of the proposed model and assess the convergence of the spatial discretization scheme, the predicted heat transfer rates ($\dot{Q}_{\text{model}}$) were compared against the benchmark results provided by the reference software ($\dot{Q}_{\text{EVAP-COND}}$). The predictive performance of the model is evaluated and illustrated in the subsequent figures using the percentage deviation, which is defined as:

$$\text{Percentage deviation} = \frac{\dot{Q}_{\text{EVAP-COND}} - \dot{Q}_{\text{model}}}{\dot{Q}_{\text{EVAP-COND}}} \cdot 100 \quad (41)$$

As shown in Fig. 5a and b, the accuracy of the proposed model improved as the number of segments increased, gradually converging toward the EVAP-COND reference solution. The numerical solution was considered converged when the absolute difference in the computed heat transfer rate between consecutive grid refinements was less than 10 W for three consecutive steps. This 10 W threshold was selected because it represents a negligible fraction of the total heat exchanged. Based on this criterion, a 9-segment discretization was found to be sufficient to achieve a satisfactory convergence for the condenser, while the evaporator required a finer discretization of 14 segments. For reference, the total heat transfer rates obtained from EVAP-COND were 2796 W (consisting of 821 W latent and 1974 W sensible heat) for the evaporator and 3150 W for the condenser.

At convergence, the total heat transfer rate predicted for the condenser deviated by approximately 2.7% from the EVAP-COND reference data, corresponding to an absolute difference of 85 W. In contrast, under the evaporator operating condition, the proposed model underestimated the sensible heat transfer by approximately 2.4% (an absolute difference of 47 W compared to the reference) and overestimated the latent heat transfer by 6.8% (an absolute difference of 56 W). Despite these individual discrepancies, the compensating effect between the sensible and latent heat contributions limited the error on the total evaporator capacity to 0.36%, which translates to a minimal absolute difference of just 9 W from the EVAP-COND results.

For the evaporator case the larger discrepancy in latent heat was likely attributable to differences in the air-side condensation model and the use of different correlations for the heat transfer coefficients on the refrigerant sides. Specifically, EVAP-COND employs correlation [37] of Wang et al. for the air-side heat transfer coefficient, while for the refrigerant side, it utilizes the McAdams correlation [42] for single-phase flow in smooth tubes, and the Jung and Didion [43] correlation for boiling flow. To validate this hypothesis, a sensitivity analysis was conducted to assess the impact of the refrigerant-side heat transfer coefficient (HTC). As detailed in Table 2, applying an enhancement multiplier to the refrigerant-side HTC demonstrates that the predicted heat transfer rates are highly sensitive to this parameter. Notably, a minor increase in the HTC multiplier from 1.0 to 1.2 drastically reduces the latent heat deviation from 6.8% to 1.1% and improves the sensible heat underprediction from -2.9% to -0.7% . This trend confirms that the initial deviations are primarily driven by the different empirical correlations used to compute the 2P heat transfer coefficients, underscoring its critical role in accurately modeling the moisture condensation process.

It is worth noting that, for the evaporator case (Fig. 5b), when more than five segments were used (specifically, two segments in each of the

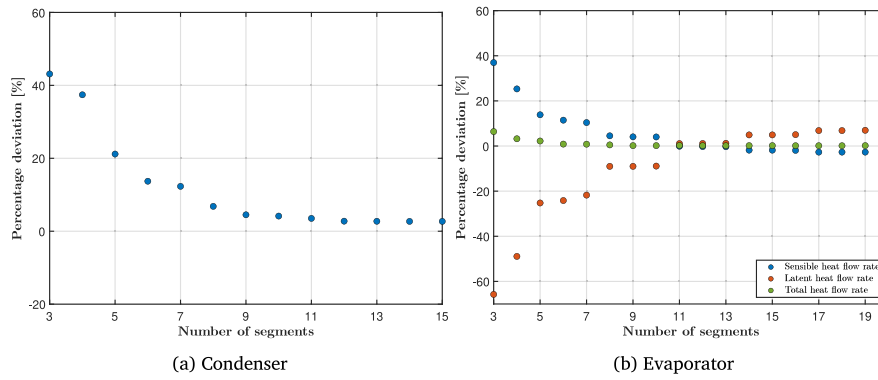


Fig. 5. Heat flow rate percentage deviation in condenser (a) and evaporator (b) operating condition.

Table 2

Effect of refrigerant-side heat transfer coefficient (HTC) enhancement on sensible, latent, and total heat transfer deviations.

Refrigerant side HTC enhancement	Percentage deviation Sensible heat [%]	Percentage deviation Latent heat [%]	Percentage deviation Total heat [%]
1	-2.9	6.8	0.4
1.1	-1.8	3.9	0.3
1.2	-0.7	1.1	0.2
1.3	-0.2	-1.2	-0.1
1.4	0.9	-3.2	0.1

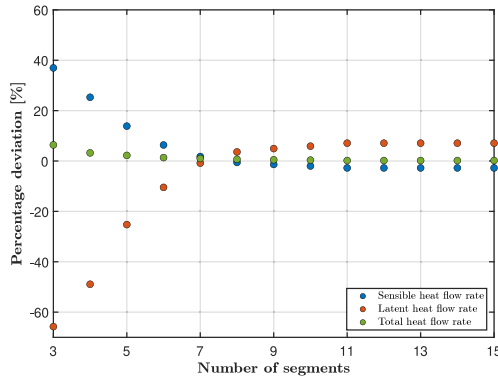


Fig. 6. Evaporator percentage deviation with respect to the refinement of the second row discretization.

first two rows and one in the third), the accuracy improved only if the discretization of the second row was further refined. In contrast, the results remained almost insensitive to the number of segments in the first and third rows. To better illustrate this, Fig. 6 shows the same convergence analysis, but with additional discretization applied exclusively to the second row, while the first and third rows were kept coarser (three segments and one segment, respectively).

This behavior is dictated by the local thermodynamics of the heat exchanger. In the specific case considered, the transition from a two-phase state to superheated vapor occurs within the second row, triggering a sharp spatial variation in the refrigerant-side heat transfer coefficient. In addition, because the inlet temperature difference between the two-phase refrigerant and the moist air remains substantial at this stage, significant sensible heat transfer persists. The coupling of these effects generates a steep wall temperature gradient along the second row, making both the air-side moisture condensation rate and the corresponding latent heat transfer highly sensitive to spatial discretization. Physically, a similar requirement for refinement arises in the first row when the refrigerant mass flow rate is low relative to the moist-air flow rate. Conversely, while such conditions could theoretically occur in the third row

if the refrigerant mass flow rate were disproportionately high, the reduced inlet temperature difference at that stage typically limits the wall temperature gradient. Consequently, further refinement in the third row is often computationally unjustified.

These findings suggest that a non-uniform discretization strategy is more efficient for the evaporator under analysis. By selectively refining only the first and second rows, a 9-segment discretization (configured as 3, 5, and 1 segments for the respective rows) was found to be sufficient to achieve a satisfactory convergence error. This optimized scheme reduced the computational cost by approximately 36% compared to a uniform 14-segment discretization without compromising the accuracy too much. Although the specific 36% reduction is tied to this particular operating condition and the localized phase transition, the underlying strategy of non-uniform discretization is highly generalizable. Selectively refining the grid only in regions with sharp thermodynamic gradients provides a consistent method to optimize computational efficiency across different operating scenarios.

In terms of computational performance, EVAP-COND 5.0 required 5.8 s to reach the steady-state solution for both the condenser and evaporator cases. Notably, attempting to reduce the computational overhead by selecting the coarsest spatial discretization permitted by the reference software (one segment per tube) yielded a negligible execution time reduction of only 0.2 s for the specific configurations analyzed. Conversely, the proposed converged models—configured with 9 segments for the condenser and 9 segments for the evaporator (using the optimized discretization scheme)—simulated 2000 s (enough to reach a steady-state solution) in approximately 1.20 s for the condenser case and 1.77 s for the evaporator case, resulting in a run-to-simulation time ratio of about 0.6 and 0.885 ms per simulated second, respectively. Simulations were performed using an Intel® Core Ultra 7-265K processor.

While the two software environments utilize different solver and implementations, this comparison provides a practical reference against an established industry standard. Because dynamic models inherently require significantly higher computational effort to solve time-dependent balances, reaching a steady-state solution faster than a dedicated steady-state tool underscores the exceptional efficiency of the proposed framework and its suitability for real-time, system-level simulations.

3.2. Experimental comparison

To complement the numerical comparison against the EVAP-COND software, the proposed heat exchanger model was further assessed using experimental data. It is worth noting that this model was originally developed and integrated into a system-level simulation of an Exhaust Air Heat Pump (EAHP) in our previous study [44]. While that framework was successfully validated at the macroscopic level, demonstrating deviations within $\pm 10\%$ for the overall capacity, the present analysis specifically isolates the evaporator and condenser components. To accurately evaluate their individual thermal contributions, the exact experimental

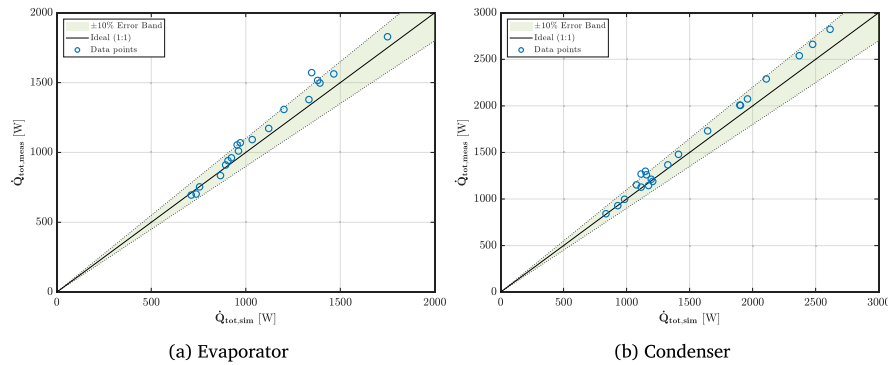


Fig. 7. Comparison between simulated and measured total heat flow rates for the evaporator (a) and condenser (b).

Table 3

MAE, MAPE, RMSE and RMSPE for total heat flow rates.

	Evaporator [W]	Condenser [W]
MAE (MAPE)	76 (6%)	87 (5%)
RMSE (RMSPE)	94 (7%)	109 (6%)

boundary conditions recorded on both the air side and the refrigerant side during the steady-state summer campaign were extracted and directly imposed on the standalone component models. Specifically, 21 distinct test conditions were compared to rigorously evaluate the performance of the components under varying operational scenarios. The full details of this experimental setup and data acquisition are extensively discussed in [44]. To ensure transparency and reproducibility, the corresponding datasets have been made available in a public repository [26].

Fig. 7 illustrates the comparison between the simulated and measured total heat transfer rates for both components under these specific conditions. As shown, the numerical predictions align closely with the experimental measurements, with the majority of the data points falling safely within the $\pm 10\%$ error band. This visual agreement is quantitatively supported by the statistical metrics summarized in Table 3. The low magnitude of the Mean Absolute Error (MAE), Mean Absolute Percentage Error (MAPE), Root Mean Square Error (RMSE) and Root Mean Square Percentage Error (RMSPE) for both heat exchangers, along with their comparable magnitudes, indicates a consistent predictive capability without significant local outliers. Such a level of accuracy is particularly remarkable when considering the inherent uncertainties associated with the empirical heat transfer correlations utilized in the model, coupled with the inevitable propagation of experimental measurement errors. For instance, comparable predictive accuracies were reported when validating the well-established EVAP-COND simulation tool against experimental data, with deviations of approximately 7% [45] and 8% [46]. In the same study, Avanzo et al. [45] also evaluated EVSIM [47], another finite-volume simulation software, observing maximum errors of up to 12%. Consequently, these results robustly confirm the physical consistency and empirical reliability of the proposed model.

4. Conclusion

This study introduced a dynamic, segment-by-segment fin-and-tube heat exchanger model designed to reconcile high-fidelity spatial discretization with the computational efficiency required for real-time, system-level simulations. Based on numerical and experimental assessments, the main conclusions of this work are summarized as follows:

- The white-box open-source architecture, developed natively in the Simscape language, provides a highly customizable alternative to conventional closed-source software. This framework allows the di-

rect coupling of detailed heat exchanger models with advanced control algorithms, eliminating the need for complex co-simulation interfaces.

- A non-uniform spatial discretization scheme proved to be an effective strategy to reduce computational cost. Grid convergence analysis indicated that refining the segments exclusively in regions with steep thermodynamic gradients, such as those experiencing phase transitions, decreased the computational effort by approximately 36% for the evaporator case without noticeable loss in accuracy.
- The model demonstrated real-time simulation capabilities due to optimized numerical treatments. It completed a 2000-second transient simulation in under 2 s, achieving run-to-simulation time ratios below 1 millisecond. This execution speed confirms its suitability for hardware-in-the-loop testing and system-level digital twins.
- Despite its low computational cost, the low-order formulation maintained high predictive accuracy. The model showed strong agreement with the EVAP-COND 5.0 reference software, predicting the total capacity with a maximum deviation of 2.7% for the condenser and 0.36% for the evaporator. Furthermore, deviations from experimental data consistently remained within a $\pm 10\%$ error band, showing low mean absolute and root mean square percentage errors.

Future work will focus on the experimental validation of the model under transient conditions and the integration of a frost formation model to extend its applicability to winter operation under frosting conditions.

CRedit authorship contribution statement

Roberto Sedoni: Writing – original draft, Visualization, Validation, Methodology, Investigation, Formal analysis, Data curation, Conceptualization; **Paolo E. Santangelo:** Writing – review & editing, Supervision, Project administration, Methodology, Funding acquisition, Conceptualization; **Diego Angeli:** Writing – review & editing, Supervision, Project administration, Methodology, Funding acquisition, Conceptualization.

Declaration of competing interest

The authors declare that they have no known competing financial interests or personal relationships that could have appeared to influence the work reported in this paper.

Acknowledgments

We express our gratitude to Zehnder Group - Competence Center Campogalliano (MO) for their assistance and the allocation of necessary resources throughout this research endeavor.

Data availability

Data will be made available on request.

References

- [1] A.C. Ispir, G.O. Rodriguez, W. De Vries, M. Speetjens, Digital twin development of a full-scale industrial heat pump, *Appl. Therm. Eng.* 269 (2025) 125921. <https://doi.org/10.1016/j.applthermaleng.2025.125921>
- [2] B. Guo, X. Huang, Q. Li, Y. Fan, Y. Yang, Y. Wang, J. He, A complete ASHP-based HVAC hybrid model for building energy retrofits via two-stage event-triggered model predictive control, *J. Build. Eng.* 100 (2025) 111624. <https://doi.org/10.1016/j.jobe.2024.111624>
- [3] L. Tang, H. Xie, Y. Wang, Z. Xu, Deeply flexible commercial building HVAC system control: a physics-aware deep learning-embedded MPC approach, *Appl. Energy* 388 (2025) 125631. <https://doi.org/10.1016/j.apenergy.2025.125631>
- [4] Q. Si, J. Wei, Y. Li, H. Cai, Optimization for the model predictive control of building HVAC system and experimental verification, *Buildings* 12 (10) (2022) 1602. <https://doi.org/10.3390/buildings12101602>
- [5] X.-D. He, S. Liu, H.H. Asada, Modeling of vapor compression cycles for multivariable feedback control of HVAC systems, *J. Dyn. Syst. Meas. Control* 119 (2) (1997) 183–191. <https://doi.org/10.1115/1.2801231>
- [6] M. Gräber, N.C. Strupp, W. Tegethoff, Moving boundary heat exchanger model and validation procedure, in: *EUROSIM Congress on Modelling and Simulation*, 2010.
- [7] D. Kim, D. Ziviani, J.E. Braun, E.A. Groll, A moving boundary modeling approach for heat exchangers with binary mixtures, *Energy Procedia* 129 (2017) 466–473. <https://doi.org/10.1016/j.egypro.2017.09.161>
- [8] P. Mithraratne, N.E. Wijesundera, T.Y. Bong, Dynamic simulation of a thermostatically controlled counter-flow evaporator, *Int. J. Refrig.* 23 (3) (2000) 174–189. [https://doi.org/10.1016/s0140-7007\(99\)00048-1](https://doi.org/10.1016/s0140-7007(99)00048-1)
- [9] W.-D. Gruhle, R. Isermann, Modeling and control of a refrigerant evaporator, *J. Dyn. Syst. Meas. Control* (1985). <https://doi.org/10.1115/1.3140728>
- [10] M. Giovannini, M. Lorenzini, Switching criteria analysis for a condenser moving boundary model 2685 (2024) 012040. <https://doi.org/10.1088/1742-6596/2685/1/012040>
- [11] B.P. Rasmussen, B. Shenoy, Dynamic modeling for vapor compression systems—Part II: Simulation tutorial, *HVAC&R Res.* 18 (5) (2012) 956–973. <https://doi.org/10.1080/10789669.2011.582917>
- [12] R. Sedoni, P.E. Santangelo, D. Angeli, M. Romani, L. Fioravanti, An analytical model of evaporator performance under frosting conditions for lumped-parameter simulations of heat pumps 2940 (2025) 012015. <https://doi.org/10.1088/1742-6596/2940/1/012015>
- [13] H. Qiao, V. Aute, R. Radermacher, Dynamic modeling and characteristic analysis of a two-stage vapor injection heat pump system under frosting conditions, *Int. J. Refrig.* 84 (2017) 181–197. <https://doi.org/10.1016/j.ijrefrig.2017.08.020>
- [14] A. Desideri, B. Dechesne, J. Wronski, M. Van den Broek, S. Gusev, V. Lemort, S. Quoilin, Comparison of moving boundary and finite-volume heat exchanger models in the modelica language, *Energies* 9 (5) (2016) 339. <https://doi.org/10.3390/en9050339>
- [15] Center for Environmental Energy Engineering, CoilDesigner®, 2025, (Software). <https://optimizedthermalsystems.com/coildesigner/>
- [16] P.A. Domanski, D.A. Yashar, J. Wojtusiak, *EVAP-COND; Simulation Models for Finned-Tube Heat Exchangers with Circuitry Optimization*, Version 5.0, National Institute of Standards and Technology, Gaithersburg, MD, 2021.
- [17] P. Jin, J. Feng, Design and performance analysis of a finned tube heat exchanger, in: *Journal of Physics: Conference Series*, 3041, IOP Publishing, 2025, p. 012017. <https://doi.org/10.1088/1742-6596/3041/1/012017>
- [18] J. Jabbour, S. Russeil, M. Mobtil, D. Bougeard, M.-F. Lacrampe, P. Krawczak, High performance finned-tube heat exchangers based on filled polymer, *Appl. Therm. Eng.* 155 (2019) 620–630. <https://doi.org/10.1016/j.applthermaleng.2019.04.036>
- [19] R. Yu, Y. Chen, Study on dehumidification and heat exchange performance of air source heat pump drying system, in: *2025 6th International Conference on Energy Power and Automation Engineering (ICEPAE)*, IEEE, 2025, pp. 356–362. <https://doi.org/10.1109/icepae66132.2025.11275714>
- [20] S.A.A. Mehrjardi, A. Khademi, S.M.M. Safavi, Performance increasement in shell-and-tube heat exchangers reinforced with dimpled tubes: a correlation-based approach, *Int. J. Heat Mass Transf.* 226 (2024) 125489. <https://doi.org/10.1016/j.ijheatmasstransfer.2024.125489>
- [21] G. Priyadarshi, C. Murali, S. Agarwal, B.K. Naik, Parametric investigation and optimization of phase change material-based thermal energy storage integrated desiccant coated energy exchanger through physics informed neural networks oriented deep learning approach, *J. Energy Storage* 80 (2024) 110231. <https://doi.org/10.1016/j.est.2023.110231>
- [22] A.H. Abdulkarim, O. Nooruldeen, A. Ghareeb, D. Akgül, H. Mercan, A.S. Dalkilic, Physics-guided machine learning and deep neural networks for accurate and robust heat transfer predictions in shell and helical coil heat exchangers, *Arabian J. Sci. Eng.* 51 (7) (2026) 9253–9280. <https://doi.org/10.1007/s13369-025-10601-3>
- [23] V. Kalpana, G.J. Sujana, K. Thyagarajan, R. Lalitha, V. Talasila, M.M. Jadhav, Enhancing heat transfer coefficient predictions in complex geometries through hybrid machine learning approaches, *Therm. Sci. Eng. Prog.* 55 (2024) 103017. <https://doi.org/10.1016/j.tsep.2024.103017>
- [24] C. Bhattacharya, A. Chakrabarty, C. Laughman, H. Qiao, Modeling nonlinear heat exchanger dynamics with convolutional recurrent networks, *IFAC-PapersOnLine* 55 (37) (2022) 99–106. <https://doi.org/10.1016/j.ifacol.2022.11.168>
- [25] X.-Y. Liang, B. Zhang, C.-L. Zhang, Physics-informed deep residual neural network for finned-tube evaporator performance prediction, *Energy* 302 (2024) 131822. <https://doi.org/10.1016/j.energy.2024.131822>
- [26] R. Sedoni, Research data and files for the paper "An accurate low-order model of a fin-and-tube heat exchanger for the dynamic simulation of heat pumps", 2026, (https://git.unimore.it/225791/research_data_re_paper). Repository GitLab.
- [27] V. Gnienlinski, On heat transfer in tubes, *Int. J. Heat Mass Transf.* 63 (2013) 134–140. <https://doi.org/10.1016/j.ijheatmasstransfer.2013.04.015>
- [28] A. Cavallini, R. Zecchin, A dimensionless correlation for heat transfer in forced convection condensation, in: *International Heat Transfer Conference Digital Library*, Begel House Inc., 1974. <https://doi.org/10.1615/ihct5.1220>
- [29] M.M. Shah, A new correlation for heat transfer during boiling flow through pipes, *ASHRAE Trans.* 82 (2) (1976) 66–86. <https://doi.org/10.1115/imece2018-86589>
- [30] D.P. Sekulic, R.K. Shah, *Fundamentals of Heat Exchanger Design*, John Wiley & Sons, 2023.
- [31] R. Sánta, The analysis of two-phase condensation heat transfer models based on the comparison of the boundary condition, *Acta Polytech. Hung.* 9 (6) (2012) 167–180. <https://doi.org/10.12700/aph.9.6.2012.6.11>
- [32] A. Filip, F. Baltaretu, R.-M. Damian, Comparison of two-phase pressure drop models for condensing flows in horizontal tubes, *Model. Civil Environ. Eng.* (4) (2014) 22. <https://doi.org/10.2478/mmce-2014-0019>
- [33] S.E. Haaland, Simple and explicit formulas for the friction factor in turbulent pipe flow, *J. Fluids Eng.* 105 (1) (1983) 89–90. https://asmedigitalcollection.asme.org/fluidsengineering/article-pdf/105/1/89/5711465/89_1.pdf <https://doi.org/10.1115/1.3240948>
- [34] W.R. Lockhart, Proposed correlation of data for isothermal two-phase, two-component flow in pipes, *Chem. Eng. Prog.* 45 (1) (1949) 39–48.
- [35] W.M. Kays, A.L. London, *Compact Heat Exchangers*, Krieger Publishing Company, 1998.
- [36] C.-C. Wang, K.-Y. Chi, C.-J. Chang, Heat transfer and friction characteristics of plain fin-and-tube heat exchangers, Part II: correlation, *Int. J. Heat Mass Transf.* 43 (15) (2000) 2693–2700. [https://doi.org/10.1016/S0017-9310\(99\)00333-6](https://doi.org/10.1016/S0017-9310(99)00333-6)
- [37] C.-C. Wang, C.-J. Lee, C.-T. Chang, S.-P. Lin, Heat transfer and friction correlation for compact louvered fin-and-tube heat exchangers, *Int. J. Heat Mass Transf.* 42 (11) (1999) 1945–1956. [https://doi.org/10.1016/S0017-9310\(98\)00302-0](https://doi.org/10.1016/S0017-9310(98)00302-0)
- [38] J. Min, M. Su, Performance analysis of a membrane-based energy recovery ventilator: effects of membrane spacing and thickness on the ventilator performance, *Appl. Therm. Eng.* 30 (8–9) (2010) 991–997. <https://doi.org/10.1016/j.applthermaleng.2010.01.010>
- [39] H. Nemati, S. Samivand, Simple correlation to evaluate efficiency of annular elliptical fin circumscribing circular tube, *Arabian J. Sci. Eng.* 39 (12) (2014) 9181–9186. <https://doi.org/10.1007/s13369-014-1474-z>
- [40] F. Suárez, S.D. Keegan, N.J. Mariani, G.F. Barreto, A novel one-dimensional model to predict fin efficiency of continuous fin-tube heat exchangers, *Appl. Therm. Eng.* 149 (2019) 1192–1202. <https://doi.org/10.1016/j.applthermaleng.2018.12.125>
- [41] S. Padhmanabhan, Study of Frost Growth on Heat Exchangers used as Outdoor Coils in Air Source Heat Pump Systems, Ph.D. thesis, Oklahoma State University, 2011.
- [42] A.-C. American Society of Heating, Refrigerating and Engineers, 2009 *ASHRAE Handbook: Fundamentals*, ASHRAE, Atlanta, GA, Atlanta, GA, 2009.
- [43] Horizontal flow boiling heat transfer experiments with a mixture of R22/R114, *Int. J. Heat Mass Transf.* 32 (1) (1989) 131–145. [https://doi.org/10.1016/0017-9310\(89\)90097-5](https://doi.org/10.1016/0017-9310(89)90097-5)
- [44] R. Sedoni, Modellazione E Analisi Sperimentale Di Sistemi Avanzati Per La Climatizzazione Di Ambienti Confinati, Ph.D. thesis, Università degli studi di Modena e Reggio Emilia, 2026.
- [45] R.H. Avanço, R. Perussi, L.C. Gómez, H.A. Navarro, Numerical analysis of performance in a finned-tube evaporator, in: *Proceedings of the 13th Brazilian Congress of Thermal Sciences and Engineering (ENCIT 2010)*, ABCM, Uberlandia, MG, Brazil 2010. <https://doi.org/10.13140/RG.2.1.3604.3281>
- [46] N. Jing, Y. Xia, Q. Ding, Y. Chen, Z. Wang, X. Zhang, Simulation and optimization study on the performance of fin-and-tube heat exchanger, *Sustainability* 15 (15) (2023) 11587. <https://doi.org/10.3390/su151511587>
- [47] P.A. Domanski, P.A. Domanski, *EVSIM: An Evaporator Simulation Model Accounting for Refrigerant and one Dimensional Air Distribution*, US Department of Commerce, National Institute of Standards and Technology, 1989.