

CFD Modelling of Di-Phasic Refrigerant Inside an Aircraft Skin Heat Exchanger as a Condenser for Hybrid-Electric Regional Aircraft [†]

Iván González-Nieves * , Andrés Felgueroso-Rodríguez , Miguel Díaz-Barja and Jorge García-Rodríguez

Airbus Defence and Space S.A.U., 28906 Getafe, Madrid, Spain; andres.felgueroso@airbus.com (A.F.-R.); miguel.diaz.barja@airbus.com (M.D.-B.); jorge.r.garcia@airbus.com (J.G.-R.)

* Correspondence: ivan.gonzalez.nieves@airbus.com

[†] Presented at the 15th EASN International Conference, Madrid, Spain, 14–17 October 2025.

Abstract

The development of future electrical aircraft, such as the Hybrid-Electric Regional Aircraft (HERA) platform, presents challenging cooling demands due to the heat generated by electric powerplants, fuel cells and power electronics. Traditional heat exchangers in ram air channels may not be sufficient, necessitating alternative solutions like Skin Heat Exchangers (SHXs) to enhance heat transfer and reduce parasitic drag. Aircraft drag reduction and efficiency increase are expected with the integration of SHXs in two-phase cooling systems. This study employs Computational Fluid Dynamics (CFD) models, specifically the Volume of Fluid (VOF) multiphase model together with the Lee model, to simulate the condensation process of two Hydrofluoroolefin (HFO) refrigerants in SHX channels (R1233zd(E) and R1234yf). An analytical model based on empirical equations is used to preliminarily correlate and validate the CFD results, showing deviations below 15%. The simulations reveal distinct flow behaviours for each refrigerant, influenced by the differences in liquid and gas densities. The study also establishes a basis for understanding and selecting the inverse of the relaxation time coefficient, which is crucial for multiphase CFD modelling. The CFD models used in this article could be of great importance for future SHX design optimization.

Keywords: skin heat exchanger; condensation; aircraft; multiphase; refrigerant; CFD; channel; VOF

1. Introduction

The aviation industry is undergoing a significant transformation towards electrification, driven by the need to reduce emissions and improve energy efficiency. This shift is particularly evident in the development of Hybrid-Electric Regional Aircraft [1] (HERA), which integrate electric propulsion systems with conventional powerplants. One of the critical challenges in this transition is the effective thermal management of high-density thermal loads generated by state-of-the-art power electronics and hydrogen fuel cells. Traditional ram air channels, while effective, contribute to increased aircraft drag, thereby compromising overall efficiency [2].

To address this challenge, innovative cooling solutions are being explored, one of which is the integration of skin heat exchangers that utilize the high latent heat of condensation of a di-phasic refrigerant as part of a two-phase cooling system, e.g., the Vapour Cycle System (VCS) and Mechanically Pumped Loop (MPL). These heat exchangers, attached



Academic Editors: Spiros Pantelakis,
Andreas Strohmayer and Gustavo
Alonso

Published: 13 May 2026

Copyright: © 2026 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 \(CC BY\)](https://creativecommons.org/licenses/by/4.0/) license.

to the inner surface of the aircraft skin, offer a promising alternative for dissipating heat during typical cruise scenarios without significantly impacting the aircraft's aerodynamic performance. Understanding the flow behaviour and heat transfer characteristics within these channels is crucial for optimizing their design and performance [3–5].

Computational Fluid Dynamics (CFD) plays a pivotal role in this endeavour, providing detailed insights into the complex interactions between the refrigerant phases and the heat exchanger surfaces. CFD simulations enable the prediction of flow patterns, pressure drops, and heat transfer rates, which are essential for the efficient design of these systems. Previous studies, such as those by Wang et al. [6] and Yao et al. [7], have demonstrated the efficacy of CFD in modelling two-phase flows and heat transfer in various applications, highlighting its potential for advancing thermal management solutions in aviation. Additionally, research by Apanasevich et al. [8] has shown the importance of the accurate modelling of condensation processes in enhancing the performance of heat exchangers.

This paper presents a CFD modelling approach to investigate the condensation of a di-phasic refrigerant within the channels of an aircraft skin heat exchanger designed for HERA. The study aims to enhance the understanding of flow behaviour and heat transfer mechanisms, ultimately contributing to the development of more efficient and sustainable thermal management systems for electrified aircraft. By leveraging CFD, this research seeks to optimize the design of skin heat exchangers, reducing reliance on traditional ram air intakes and paving the way for more energy-efficient and environmentally friendly aviation technologies [9].

2. Materials and Methods

2.1. Geometry Description

The Skin Heat Exchanger (SHX) concept currently under development employs a parallel channel configuration attached to the mentioned inner surface of the aircraft skin. This configuration includes a collector at each end, which serves to join the inlet and outlet pipes, respectively, to the rest of the cooling circuit. As the SHX functions as a condenser heat exchanger, the refrigerant enters in a gaseous state, condensates, and exits in a liquid state while transferring heat to the external air (Figure 1a). That operational capability enables the SHX to be integrated as the condenser in various two-phase cooling systems already integrated on aircraft platforms, such as the VCS (Figure 1b) and MPL. It is also important to note that the SHX operates in a counter-flow configuration, wherein the refrigerant circulates in the opposite direction to the external air.

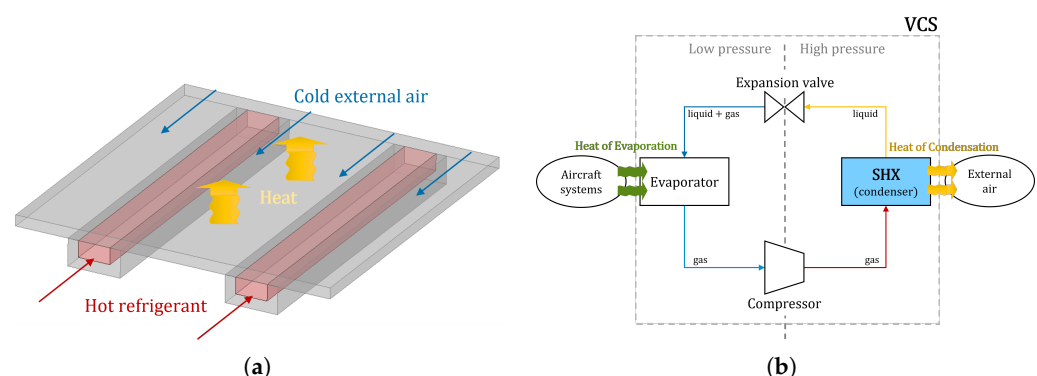


Figure 1. (a) SHX flow and heat direction. (b) VCS scheme with SHX integrated as condenser.

These CFD studies focus exclusively on the phase change of the refrigerant, neglecting conduction through surfaces and convection from external air. The target design intended for aircraft installation comprises a total of seventeen parallel channels, with a total SHX width of 0.5 m and an approximate length of 1 m (the exact length being dependent on

the specific refrigerant used). Assuming the refrigerant behaviour during condensation is similar within all channels, the geometry has been simplified for CFD simulation purposes. Specifically, only half of a channel is modelled, with symmetry conditions applied to reduce computational cost (Figure 2). Consequently, only half of the volume occupied by the refrigerant inside the channels is used in CFD simulations.

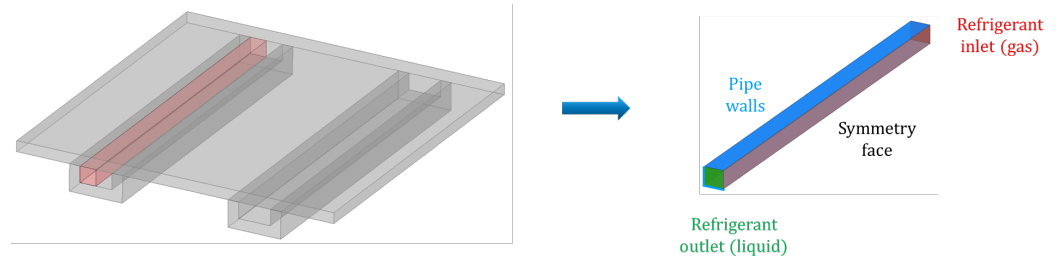


Figure 2. Channel geometry simplification for CFD.

2.2. Boundary and Simulation Conditions

The half-channel geometry is characterized by a height and width of 3 millimetres each, with the length determined by the specific refrigerant employed. As illustrated in Figure 2, the boundary conditions (BCs) for the simulation have been defined as follows: mass flow at the inlet, a relative pressure outlet set to zero, symmetry conditions applied to the symmetry plane, and no-slip conditions imposed on the pipe/channel walls.

Two HFO (hydrofluoroolefin) refrigerants, R1234yf and R1233zd(E), have been simulated and analysed independently. Both refrigerants share certain simulation characteristics, while others are specific to each of them. The particular conditions of the simulated cases are as follows:

- Inlet refrigerant temperature: 75 °C (vapour).
- Condensation temperature: 70 °C (two-phase).
- Outlet temperature (expected): 65 °C (liquid).
- Target heat dissipation: 3250 W (all 17 channels); 95.6 W (each half-channel).
- Wall temperature: Dependent on each refrigerant. As an initial simplification the wall temperature was fixed to simulate heat transfer towards the exterior air, corresponding to cruise at a 25 kft altitude, under ISA conditions and at a 300 KTAS velocity.
- Refrigerant mass flow:

$$\dot{m} = \frac{Q}{h_{in} - h_{out}} = \frac{Q}{h_{gas} - h_{liquid}} \quad (1)$$

with $\dot{m}$ being the mass flow rate of the refrigerant in kg/s, Q the target heat dissipation during the condensation process in W, h_{in}/h_{gas} the specific enthalpy of the refrigerant at the inlet of the SHX in the gaseous state and h_{out}/h_{liquid} the specific enthalpy of the refrigerant at the SHX outlet as liquid.

Prior to conducting the CFD modelling, and out of the scope of this manuscript's work, a numerical/analytical model was developed to estimate the performance of the SHX and to size preliminary prototypes (Felgueroso et al. [5]). This model implements empirical correlations and enables the estimation of the complete heat balance among the external air, SHX walls, and refrigerant inside the channels. It is assumed that the channel wall temperature varies solely along the length axis but remains constant at any given cross-sectional area of the channels. Consequently, the wall temperature results obtained from the numerical model are utilized as boundary conditions for the CFD modelling in order to simulate only the refrigerant condensation in the channel interior.

2.3. Numerical Methods and Models

As with most CFD simulations in industry, while detailed information about each particle in a multiphase flow can be obtained, the practical interest primarily lies in the quantities derived after applying some form of averaging to simplify the data. This suggests the use of Reynolds Averaged Navier–Stokes (RANS) equations for the CFD multiphase models as well (Prosperetti and Tryggvason [10]).

Regarding the turbulence model, k-epsilon is employed to predict the turbulent flow behaviour. The decision was focused on reducing computational cost by taking advantage of wall functions, and also the well-known robustness of this turbulence model.

2.3.1. Multiphase Model

Moreover, it is essential to utilize an appropriate multiphase model capable of handling two distinct fluids and accurately simulating their interaction and interface within the computational domain. In this condensation process, the software treats both the gas and liquid refrigerant as separate fluids. The primary multiphase modelling options available are the Volume of Fluid (VOF) model, the Mixture model and the Eulerian model [11].

Several studies in the literature provide arguments for particular multiphase model selection. Sarkar [12] and Mohammed et al. [13]’s work on CFD multiphase models and the empirical equations employed in the analytical model (Felgueroso et al. [5]) correspondent to annular/film condensation flow led to the decision of using the VOF model for this study, designed for two or more immiscible fluids where the interface is of interest [11].

2.3.2. Lee Model

Once the multiphase model is selected, a specific sub-model must be chosen and adjusted to characterize the mass transfer mechanism between the phases. For condensation processes, the applicable sub-model is called the evaporation–condensation model, in which phase change occurs as vapour transfers mass to the liquid (vapour condensates to form liquid), thereby reducing the vapour mass and increasing the liquid mass. In the context of the VOF model, the Lee model is the available sub-model to define the mass transfer mechanism.

The mass transfer between vapour and liquid during condensation (or from liquid to vapour in the case of evaporation) is governed by the vapour transport equation:

$$\frac{\delta}{\delta t}(\alpha_v \rho_v) + \nabla \cdot (\alpha_v \rho_v \vec{V}_v) = \dot{m}_{lv} - \dot{m}_{vl} \quad (2)$$

where α_v is the vapour volume fraction (-), ρ_v is the vapour density (in kg/m³), $\vec{V}_v$ is the vapour phase velocity (in m/s), and $\dot{m}_{lv}$ and $\dot{m}_{vl}$ are the mass transfer rate due to evaporation and condensation respectively (in kg/m³s).

In this case, with the Lee model, condensation of the vapour phase starts when the mass transfer rate of condensation is higher than zero ($\dot{m}_{vl} > 0$). This variable is defined as:

$$\dot{m}_{vl} = \text{coeff} \times \alpha_v \rho_v \frac{T_{\text{sat}} - T_v}{T_{\text{sat}}} \quad (3)$$

Here, T_{sat} is the saturation/condensation temperature (in °C), T_v the vapour temperature (in °C), and coeff the coefficient that represents the inverse of the relaxation time (in s⁻¹). It is notable that the mass transfer rate becomes positive when $T_v < T_{\text{sat}}$, since coeff is always positive. This is the point at which condensation begins.

The coefficient can be calculated; nevertheless, most of the reviewed literature, including Holešová et al. [14], recommend fine tuning it to match experimental data and validate the model. The higher the coefficient, the higher the mass transfer rate. If the

coefficient is too low, in order for the mass transfer rate to be enough for phase change to happen, the difference $T_{\text{sat}} - T_v$ needs to be high, which means that the vapour does not condensate at T_{sat} but at a lower temperature. Therefore, in the context of CFD modelling, it is advantageous to employ the highest possible coeff to ensure that condensation occurs when $T_v \approx T_{\text{sat}}$. This approach aims to accurately represent the condensation phenomena in the CFD model, closely mirroring the real process behaviour.

The coefficient was increased until the solution diverged, and then slightly reduced until it converged again. It was observed that, for low coeff values, the refrigerant did not completely condensate along the channel, but showed a partial condensation at a lower T_{sat} than desired. Thus, the coefficient was maximized and the condensation temperature was maintained as desired. It should be noted that the simulation is transient and, initially, the channel is empty of refrigerant, introduced as vapour. The simulation is stopped when the refrigerant completely condenses and exits the channel as liquid [15].

2.4. Mesh

A mesh independence study was performed to ensure that the solution was not affected by the utilized mesh. The number of mesh elements was modified in all axis directions to adjust the mesh characteristics in both channel section and length. Four mesh sizes were analysed: 100 k, 225 k, 576 k and 1152 k elements.

The representation of the length to complete vapour condensation (Figure 3a) and refrigerant temperature evolution along the channel length (Figure 3b) for R1233zd(E) for each mesh element size show similar results for the 576k element mesh and 1152 k element mesh. Thus, as a good compromise between solution and computational cost, the mesh with 576,000 elements was selected to evaluate the refrigerant condensation process and it was employed in the simulations of this work.

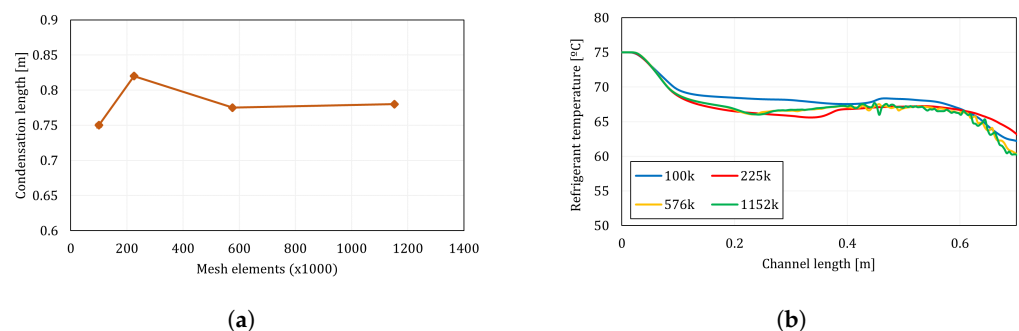


Figure 3. (a) Representation of condensation length for each mesh element. (b) Representation of refrigerant temperature evolution along the channel length for each mesh element size.

3. Results

The CFD simulation results for the R1233zd(E) and R1234yf condensation process within the SHX channel can be evaluated in various ways. The focus has primarily been on the liquid formation characteristics, vapour distribution during condensation and temperature evolution.

Figure 4 shows the evolution of the vapour volume fraction along the channel length for R1233zd(E) and R1234yf, respectively. The refrigerant flow is from right to left. Several lines are illustrated in both cases, representing different corners of the channel section, all at the walls, except for the centre position. At the inlet, the refrigerant is 100% vapour, and its quantity decreases along the channel as it condenses. For R1233zd(E), condensation takes more time and primarily occurs at the end of the channel, while for R1234yf, it is more continuous, mainly due to the density difference between the refrigerant liquid and

vapour phases. Nevertheless, for both refrigerants, complete condensation is happening at the end of the channel as expected.

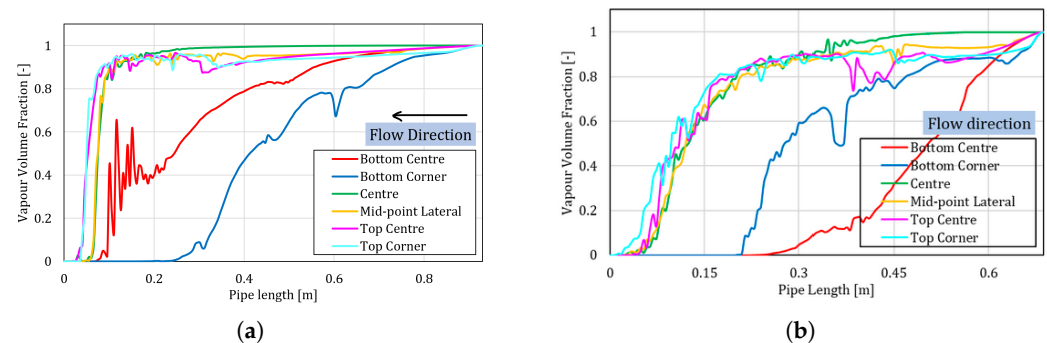


Figure 4. Vapour volume fraction evolution along channel length: (a) R1233zd(E). (b) R1234yf.

The temperature evolution along the channel for both refrigerants is displayed in Figure 5a,b. An almost constant temperature is observed during most of the channel length, corresponding to the two-phase zone where condensation is taking place. The channel centre line shown in green is distinguished among the others as it remains close to the condensation temperature value (70 °C), while the refrigerant near the walls is at a lower temperature, closer to the wall's temperature. At the channel outlets, it is noted that the refrigerant temperature in the pipe's centre is not at the target 65 °C, but at a lower temperature level, around 50 °C, which implies a slight difference with the analytical model.

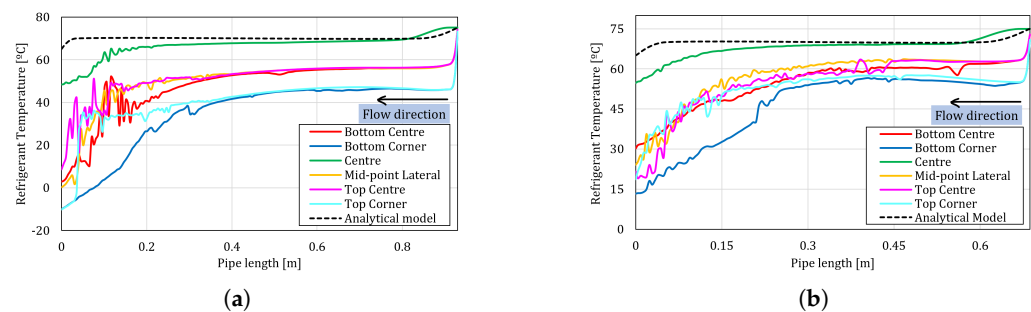


Figure 5. Refrigerant temperature evolution along channel length: (a) R1233zd(E). (b) R1234yf.

Previous results could also be observed in CFD contours. In those, it is even more clear to see the two phases' distribution along the channel, with full vapour at the inlet and complete liquid refrigerant at the outlet.

Figure 6a depicts the liquid volume fraction evolution along the channel length for the R1233zd(E), which is the opposite to the vapour volume fraction previously represented. Liquid formation (in red) starts after half of the channel length, close to the lateral walls first, and then from the bottom to the top. Figure 6b highlights the temperature evolution during condensation. The refrigerant temperature remains constant until the full liquid temperature starts to go below the saturation temperature.

On the other hand, the refrigerant R1234yf contours are represented in Figure 7a,b. From the liquid volume fraction contour, it stands out that liquid appears almost from the beginning of the channel. If the R1233zd(E) is formed suddenly, the R1234yf is more progressive along the channel. Liquid is even observed to fall from the top of the channel where it is also appearing. The R1234yf refrigerant temperature evolution is similar in regard to the R1233zd(E); nevertheless, it is appreciable that the outlet temperature stays a little higher, closer to the target outlet temperature.

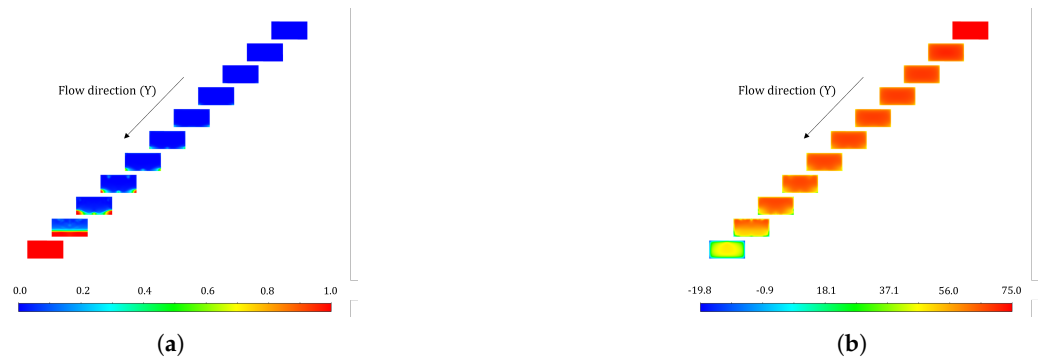


Figure 6. R1233zd(E) along the channel length: (a) Liquid volume fraction. (b) Temperature.



Figure 7. R1234yf along the channel length: (a) Liquid volume fraction. (b) Temperature.

4. Conclusions

CFD modelling allows us to better understand 3D flow patterns and behaviours that cannot be observed with analytical models, which are really interesting for the preliminary sizing of SHXs and to estimate refrigerant temperature distribution, phase change evolution, wall temperature and other factors.

Both HFO refrigerants (R1233zd(E) and R1234yf) have been simulated while condensing through the SHX channel in order to fine tune the inverse of the relaxation time coefficient (coeff) and correlate the results with the analytical model. Thus, a maximum convergent value was reached and the condensation temperature was maintained during the simulations close to the target 70 °C. Furthermore, full condensation was observed almost at the channel outlet as it is for the analytical model, with a deviation below 15% channel length in target refrigerant outlet temperature.

Even though deviation exists, a consistent CFD methodology has been prepared prior to experimental tests. The current characterization of the mesh, the relaxation time coefficient, multiphase models and others aspects will allow us to rapidly be prepared for following SHX design optimizations. As well as the analytical model, the CFD modelling will be validated and adjusted by employing dedicated experimental test data in the close future.

Author Contributions: Conceptualization, I.G.-N., A.F.-R., M.D.-B. and J.G.-R.; methodology, I.G.-N.; software, I.G.-N. and A.F.-R.; validation, I.G.-N., A.F.-R., M.D.-B. and J.G.-R.; formal analysis, I.G.-N.; investigation, I.G.-N. and A.F.-R.; resources, I.G.-N. and A.F.-R.; data curation, I.G.-N.; writing—original draft preparation, I.G.-N. and A.F.-R.; writing—review and editing, M.D.-B. and J.G.-R.; visualization, I.G.-N.; supervision, J.G.-R.; project administration, M.D.-B.; funding acquisition, M.D.-B., A.F.-R. and I.G.-N. All authors have read and agreed to the published version of the manuscript.

Funding: The project is supported by the Clean Aviation Joint Undertaking and its members. TheMa4HERA is a project co-funded by the European Union, under Grant Agreement No. 1001102008.

Institutional Review Board Statement: Not applicable.

Informed Consent Statement: Not applicable.

Data Availability Statement: The original contributions presented in this study are included in the article material. Further inquiries can be directed to the corresponding author(s).

Conflicts of Interest: All authors were employed by the Airbus Defence and Space S.A.U. company. The 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

BC	Boundary Condition
CFD	Computational Fluid Dynamics
CHT	Conjugate Heat Transfer
HERA	Hybrid-Electric Regional Aircraft
HFO	Hydrofluoroolefin
HX	Heat Exchanger
KTAS	Knots True Air Speed
MPL	Mechanically Pumped Loop
RANS	Reynolds Averaged Navier–Stokes
SHX	Skin Heat Exchanger
VCS	Vapour Cycle System
VOF	Volume of Fluid

References

1. Hybrid-Electric Regional Aircraft (HERA). 2025. Available online: <https://project-hera.eu/home> (accessed on 28 November 2025).
2. Landis, A.; Dixon-Hardy, D.; Hegg, P.; Al-Damook, M. CFD Analysis of RAM Air Flow in an Aircraft Air Conditioning System. Ph.D. Thesis, The University of Leeds, Leeds, UK, 2018. [CrossRef]
3. Lee, G.; Joo, Y.; Lee, S.U.; Kim, T.; Yu, Y.; Kim, H.G. Design optimization of heat exchanger using deep reinforcement learning. *Int. Commun. Heat Mass Transf.* **2024**, *159*, 107991. [CrossRef]
4. Han, S.; Choi, H.; Jo, I.; Lee, H. Design optimization and performance evaluation of an air-cooled heat exchanger for electric vehicle power electronics cooling. *Appl. Therm. Eng.* **2026**, *283*, 128842. [CrossRef]
5. Felgueroso, A.; González, I.; Díaz, M.; García, J. Numerical modeling of a two-phase Skin Heat Exchanger for Hybrid-Electric Regional Aircraft. In *AIAA AVIATION FORUM AND ASCEND 2024*; American Institute of Aeronautics and Astronautics: Reston, VA, USA, 2024. [CrossRef]
6. Wang, K.; Hu, C.; Cai, Y.; Li, Y.; Tang, D. Investigation of heat transfer and flow characteristics in two-phase loop thermosyphon by visualization experiments and CFD simulations. *Int. J. Heat Mass Transf.* **2023**, *203*, 123812. [CrossRef]
7. Yao, H.; Guo, L.; Liu, H.; Wang, X.; Chen, H.; Wang, Y.; Zhu, Y. Characteristics of phase-change flow and heat transfer in loop thermosyphon: Three-dimension CFD modeling and experimentation. *Case Stud. Therm. Eng.* **2022**, *35*, 102070. [CrossRef]
8. Apanasevich, P.; Lucas, D.; Beyer, M.; Szalinski, L. CFD based approach for modeling direct contact condensation heat transfer in two-phase turbulent stratified flows. *Int. J. Therm. Sci.* **2015**, *95*, 123–135. [CrossRef]
9. Coutinho, M.; Bento, D.; Souza, A.; Cruz, R.; Afonso, F.; Lau, F.; Suleman, A.; Barbosa, F.R.; Gandolfi, R.; Affonso, W.; et al. A review on the recent developments in thermal management systems for hybrid-electric aircraft. *Appl. Therm. Eng.* **2023**, *227*, 120427. [CrossRef]
10. Prosperetti, A.; Tryggvason, G. *Fundamentals of Heat Exchanger Design*, 1st ed.; Cambridge University Press: Cambridge, UK, 2009.
11. Ansys Theory Guide. 2025. Available online: https://ansyshelp.ansys.com/public/account/secured?returnurl=///Views/Secured/corp/v251/en/flu_th/flu_th_sec_mp_evap_cond.html (accessed on 17 November 2025).
12. Sarkar, S. Computational Fluid Dynamics (CFD) Modeling of Two Phase Refrigerant Flow in Evaporator Refrigerant Distribution System. In *Proceedings of the International Refrigeration and Air Conditioning Conference*; Purdue University: West Lafayette, IN, USA, 2021.

13. Mohammed, H.; Giddings, D.; Walker, G. CFD multiphase modelling of the acetone condensation and evaporation process in a horizontal circular tube. *Int. J. Heat Mass Transf.* **2019**, *134*, 1159–1170. [[CrossRef](#)]
14. Holešová, N.; Lenhard, R.; Kaduchová, K.; Malcho, M. Correlation Coefficients in Lee's Model of Multiphase Flows. In *MATEC Web of Conferences*; EDP Sciences: Les Ulis, France, 2022; p. 369. [[CrossRef](#)]
15. González, I. Modelización CFD de un Intercambiador de Calor de Superficie con Flujo Bifásico. Master's Thesis, Universitat Rovira i Virgili y Universidad Internacional de la Rioja, Tarragona, Spain, 2024.

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.