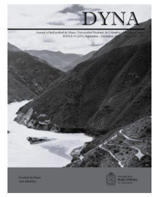
DYNA
http://dyna.medellin.unal.edu.co/



UNIVERSIDAD NACIONAL DE COLOMBIA

# Experimental analysis and computational simulation of heat transfer in a radiator

Juan Mauricio Trenado-Herrera, Crisanto Mendoza-Covarrubias, Alicia Aguilar-Corona, & Hugo Cuauhtémoc Gutiérrez-Sánchez

*Faculty of Mechanical Engineering, Michoacán University of San Nicolás de Hidalgo (UMSNH), Morelia, Michoacán, Mexico, 1597281H@umich.mx, cmendoza@umich.mx, alicia.aguilar@umich.mx, hcgsan@umich.mx*

Received: February 28th, 2025. Received in revised form: July 31st, 2025. Accepted: August 19th, 2025.

**Abstract**
This study analyzes the thermal performance of a 4.1 dm³ engine radiator through experimental tests and CFD simulations using ANSYS Fluent. The effects of materials, tube geometry, and flow conditions on heat transfer and thermal efficiency were evaluated. The results show that copper tubes enhance heat transfer by 18% but increase pressure drop by 4.44%. Additionally, increasing air velocity improves thermal efficiency by 3.74%, suggesting that specific improvements in fin design could enhance performance without increasing energy consumption. The study validates the use of CFD as a reliable tool for analyzing cooling systems in engines, benefiting the automotive industry with more efficient radiators. These improvements can be extended to hybrid and electric vehicles, as well as industrial heat exchangers, contributing to more sustainable thermal management. The main scientific contributions of this work are: (i) the experimental validation of a CFD model applied to an automotive radiator under transitional flow regime, (ii) the quantitative evaluation of the effects of copper tubes on thermal efficiency and pressure drop, and (iii) the detailed analysis of air velocity impact on heat transfer and its implications for radiator thermal design.

*Keywords*: radiator; heat transfer; CFD (Computational Fluid Dynamics); thermal efficiency; thermal design.

# Análisis experimental y simulación computacional de la transferencia de calor en un radiador.

**Resumen**
Este estudio analiza el rendimiento térmico de un radiador de motor de 4.1 dm³ mediante pruebas experimentales y simulaciones CFD en ANSYS Fluent. Se evaluaron los efectos de materiales, geometría de tubos y condiciones de flujo en la transferencia de calor y eficiencia térmica. Los resultados muestran que los tubos de cobre mejoran la transferencia de calor en un 18%, pero aumentan la caída de presión en un 4.44%. Además, incrementar la velocidad del aire mejora la eficiencia térmica en un 3.74%, lo que sugiere que ciertas mejoras en el diseño de las aletas podrían aumentar el desempeño sin afectar el consumo energético. El estudio valida el uso de CFD como herramienta confiable para el análisis de sistemas de enfriamiento en motores, beneficiando a la industria automotriz con radiadores más eficientes. Estas mejoras pueden extenderse a vehículos híbridos y eléctricos, así como a intercambiadores de calor industriales, contribuyendo a una gestión térmica más sostenible. Las principales contribuciones científicas de este trabajo son: (i) la validación experimental de un modelo CFD aplicado a un radiador automotriz en régimen de flujo transitorio, (ii) la evaluación cuantitativa del efecto de los tubos de cobre sobre la eficiencia térmica y la caída de presión, y (iii) el análisis detallado del impacto de la velocidad del aire en la transferencia de calor y sus implicaciones en el diseño térmico del radiador.

*Palabras clave*: radiador; transferencia de calor; CFD (Dinámica de Fluidos Computacional); eficiencia térmica; diseño térmico.

## 1 Introduction

The thermal performance of internal combustion engines is a critical factor in ensuring their efficiency, reliability, and compliance with increasingly strict environmental regulations. An efficient cooling system is essential to

**How to cite:** Trenado-Herrera, J.M., Crisanto Mendoza-Covarrubias, C., Aguilar-Corona, A., and Gutiérrez-Sánchez, H.C., Experimental analysis and computational simulation of heat transfer in a radiator. DYNA, (92)239, pp. 27-37, October - December, 2025.

maintain the engine temperature within an optimal range, preventing overheating and reducing fuel consumption. Among these systems, radiators play a key role by facilitating heat exchange between the coolant and the ambient air. However, radiator design faces a significant challenge: improving heat transfer without generating excessive pressure losses that could affect the performance of the cooling system. Over the past decades, research on heat transfer applied to radiators has evolved from classical theoretical models, such as the Dittus-Boelter correlations [5], to advanced simulations using computational fluid dynamics (CFD). Previous studies have shown that parameters such as tube geometry, fin distribution, and air velocity directly influence radiator thermal efficiency [1], [2]. Nevertheless, many of these investigations have relied solely on numerical simulations without detailed experimental validation, limiting their applicability in real systems. This means that, although numerical models predict thermal behavior with good approximation, the lack of experimental data hinders their practical implementation in the industry. In particular, research by Patel et al. [3] and Vajjha et al. [4] has explored heat transfer in conventional radiators but has not thoroughly evaluated the impact of pressure drop in different geometric configurations. Similarly, studies such as those by Oliet et al. [13] have analyzed thermal performance based on air velocity and tube geometry without validating their models with experimental data to confirm the accuracy of their results. These limitations leave a gap in the understanding of real radiator behavior under operational conditions. Recent literature has continued to emphasize the importance of CFD and experimental methods in enhancing radiator design. Chen et al. (2022) evaluated plate-fin compact radiators under varying inlet conditions and highlighted the role of airflow optimization in improving thermal efficiency [18]. Likewise, Kumar et al. (2021) demonstrated the accuracy of the k-ω SST model for simulating multi-tube radiators in transitional regimes, supporting its selection in this study [19]. Zhang et al. (2020) combined wind tunnel experiments with CFD simulations to validate pressure and temperature predictions in crossflow radiators [20]. These recent advances confirm the relevance of the methodological approach adopted in the present work. To address this issue, this study aims to bridge this gap by combining experimental analysis and CFD simulations to evaluate the thermal performance of a 4.1 dm³ internal combustion engine radiator. Unlike previous research, this work focuses on a detailed analysis of the influence of tube geometry and air velocity on system efficiency, validating numerical models with experimental tests conducted in a test bench. The results obtained not only provide a better understanding of radiator thermal behavior but also offer guidelines for improving its design, reducing pressure losses without compromising heat dissipation. This research is particularly relevant to the automotive industry, as it contributes to the development of more efficient cooling systems with lower energy losses and greater thermal dissipation capacity, which could reduce the environmental impact of internal combustion engines. Moreover, the proposed design improvements can be applied to the development of radiators for high-performance applications, as well as cooling systems for hybrid and electric engines, expanding their impact on the industry. The main scientific contributions of this study are: (i) the experimental validation of a CFD model applied to an automotive radiator under transitional flow regime, (ii) the quantitative evaluation of copper tube effects on thermal efficiency and pressure drop, and (iii) the detailed analysis of airflow velocity on heat transfer and its implications for radiator thermal design. Based on these contributions, the specific objectives of this study are:

(i) to evaluate the thermal behavior of an automotive radiator segment using CFD and experimental methods,
(ii) to analyze the influence of copper tube material and airflow velocity on heat transfer and pressure drop,
(iii) to validate the CFD simulation results through experimental data obtained from a custom test bench,
(iv) and to generate design recommendations based on performance indicators such as thermal efficiency and pressure loss.

## 2 Methodology

This section describes the experimental procedures and numerical simulations used to evaluate the thermal performance of a three-tube radiator with fins through computational fluid dynamics (CFD) simulations and experimental tests. The methodology focused on a detailed study of the thermal behavior of three radiator tubes, aiming to identify configurations that improve heat transfer and minimize pressure losses [16], [17]. To ensure the validity of the models and the applicability of the results, simulations were combined with experimental validation, providing a solid foundation for the research findings.

### *2.1 System selection and analysis*

A 4.1 dm³ inline six-cylinder engine with a copper-tube, aluminum-fin radiator was selected (see Fig. 1). Three tubes were analyzed to study the fluid-air interaction. Using ANSYS Fluent, the thermal behavior was modeled, and the results were validated with experimental tests, recording the coolant flow rate and temperatures (inlet: 89°C, outlet: 76°C) and air temperatures (inlet: 28°C, outlet: 39°C).

Figure 1. Radiator in the test bench.
Source: Own elaboration.

### *2.2 Simulation and validation process*

The methodological process was carried out in nine key stages. First, the problem was defined by selecting the radiator, its geometry, and establishing the operating conditions. Next, geometric modeling was developed in ANSYS, creating the computational domain. Then, the mesh was generated, with refinement in critical areas to accurately capture thermal and velocity effects. Boundary conditions were set by assigning appropriate velocities, temperatures, and thermal coefficients. The turbulence model was then selected, implementing the k-ω SST model due to its ability to handle flows with high-pressure gradients. The CFD simulation was executed by solving the conservation equations of mass, momentum, and energy. A mesh sensitivity analysis was performed to verify its independence and ensure numerical stability.

Subsequently, experimental validation was conducted by comparing the CFD results with experimental data from the test bench. Finally, a results analysis was performed, evaluating thermal efficiency, pressure drop, and proposing improvements in the radiator design.

Table 1 presents the experimental results, highlighting the cooling of the water as it flows through the radiator, contributing to maintaining the engine's ideal temperature. Additionally, a slight decrease in water pressure was observed, as well as an increase in air temperature due to heat absorption during the heat exchange process.

### *2.3 Geometric model for the radiator CFD Simulation*

The CFD model of the radiator includes three parallel tubes (6.18 mm in diameter, 748 mm in length) and zigzag fins (6 mm spacing) to maximize heat transfer. Simulated in ANSYS Fluent, it considers steady-state flow, conduction, and convection. The analysis identifies critical zones and evaluates the influence of airflow on thermal dissipation (see Fig. 2).

The selected computational domain consists of three flat tubes with their corresponding fins, which represent a simplified yet representative section of the actual radiator, which contains 84 tubes arranged in three columns of 28 tubes each.

This simplification was chosen to maintain representative thermal and aerodynamic behavior while significantly reducing computational cost. Key geometric features such as fin spacing, tube arrangement, and boundary conditions were preserved.

The inlet velocity and mass flow rate per tube were scaled to match those of the full radiator, ensuring comparable Reynolds numbers and heat transfer characteristics.

Table 1.
Experimental data on the thermal performance of the radiator in the test bench.

| Water Inlet Temperature °C | Water Outlet Temperature °C | Water Inlet Pressure, Pa | Water Outlet Pressure, Pa | Air Inlet Temperature °C | Air Outlet Temperature °C |
|---|---|---|---|---|---|
| 89 | 76 | 23248.4 | 19950.7 | 28 | 39 |

Source: Own elaboration.

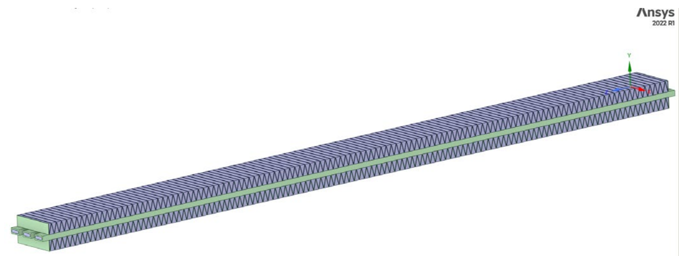



Figure 2. Geometry of the three tubes simulated using CFD.
Source: Own elaboration using ANSYS Fluent®.

This strategy has been successfully applied in previous studies [3], [17], where reduced domains were used to capture local thermal behavior with acceptable accuracy.

#### 2.3.1 Selection of the turbulence model

For the numerical simulation, the k-ω SST turbulence model was used due to its accuracy in flows with separation and high-pressure gradients, better resolution of the thermal boundary layer, and greater numerical stability. Additionally, it offers lower computational cost compared to LES and better performance than k-ε in recirculation zones.

## 3 CFD meshing of the three-tube radiator model with fins

The CFD meshing of the radiator included 6560221 elements and 4524309 nodes, with regular hexahedral elements for numerical stability. Refinements were applied at the tube-fin interfaces to capture thermal and velocity gradients accurately. The dense mesh on the fins precisely represents the air-surface interaction. Smooth transitions minimize numerical errors, ensuring a reliable thermal analysis (see Fig. 3).

### *3.1 Mesh sensitivity analysis*

A mesh sensitivity analysis was conducted by modifying the number of elements and evaluating its impact on the coolant outlet temperature and pressure drop. It was determined that with more than 6.5 million elements, variations in the results were less than 1%, ensuring accuracy without significantly increasing computational cost. This guarantees a balance between numerical accuracy and simulation efficiency.

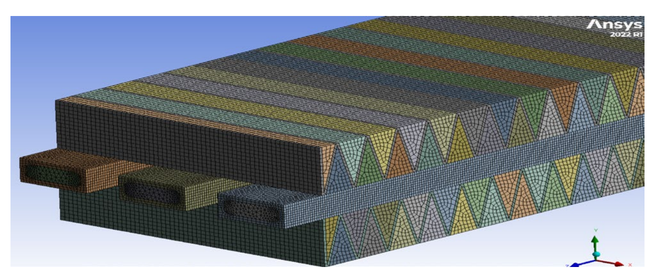



Figure 3. CFD mesh of the complete radiator, showing the three tubes and their respective fins.
Source: Own elaboration using ANSYS Fluent®.

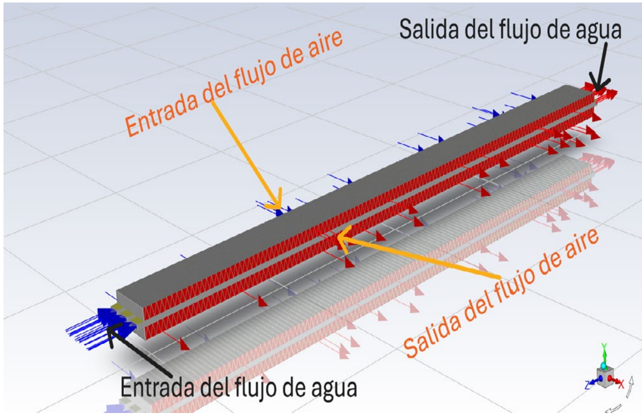



Figure 4. CFD schematic of the radiator's boundary conditions (inlet and outlet).
Source: Own elaboration using ANSYS Fluent®.

### 3.1.1 Boundary conditions

Once the mesh was generated, boundary conditions for the simulation were established. The tube and fin surfaces were modeled as thermal solids interacting with the environment, allowing heat transfer. For the water flow, an initial velocity of 0.682 m/s and an inlet temperature of 89°C were imposed. For the airflow, an initial velocity of 15.47 m/s and an inlet temperature of 28°C were assigned (see Fig. 4).

Tables 2, 3, and 4 present the inlet and outlet parameters of the water and air flow, as well as the thickness of the tubes and fins. These data are essential to ensure the accuracy of the CFD analysis.

Table 2.
Input specifications for the water flow.

| WATER FLOW INLET CONDITIONS | |
|---|---|
| Initial Water Temperature | 89°C |
| Water Velocity Through The Tube | 0.682 m/s |
| Water Inlet Pressure | 23248.4 Pa |
| Water Mass Flow Rate | 0.01916 kg/s |
| Water Convective Heat Transfer Coefficient (H) | 6447.0758 W/m$^2$*K |
| Hydraulic Diameter | 0.0046153 m |

Source: Own elaboration.

Table 3.
Input specifications for the airflow.

| AIR FLOW INLET CONDITIONS | |
|---|---|
| Initial Air Temperature | 28°C |
| airflow velocity | 15.47 m/s |
| Air Mass Flow Rate | 0.09298 kg/s |
| Air Convective Heat Transfer Coefficient (H) | 7447.45 W/m$^2$*K |
| Hydraulic Diameter | 0.0044843 m |

Source: Own elaboration.

Table 4.
Thickness specifications for the tube and fin.

| TUBE AND FIN THICKNESSES | |
|---|---|
| Tube Wall Thickness | 0.001 m |
| Fin Thickness | 0.0005 m |

Source: Own elaboration.

Table 5.
Thermophysical properties: density, viscosity, thermal conductivity, and specific heat of water.

| THERMOPHYSICAL PROPERTIES OF WATER | |
|---|---|
| Density [$kg$/m³] | 965.86 |
| Cp (Specific Heat) [ J/($kg$ − °C) ] | 4205 |
| Thermal Conductivity [W/(m − °C)] | 0.6746 |
| Viscosity [$kg$/(m − s )] | 0.0003187 |

Source: Own elaboration.

Table 6.
Thermophysical properties: density, viscosity, thermal conductivity, and specific heat of air.

| THERMOPHYSICAL PROPERTIES OF AIR | |
|---|---|
| Density [$kg$/m³] | 1.225 |
| Cp (Specific Heat) [ J/($kg$ − °C) ] | 1006.43 |
| Thermal Conductivity [W/(m − °C)] | 0.0242 |
| Viscosity [$kg$/(m − s )] | 0.000017894 |

Source: Own elaboration.

### 3.1.2 Thermophysical properties

The thermophysical properties of water and air used in the simulations are summarized in Tables 5 and 6, which include parameters such as density, viscosity, thermal conductivity, and specific heat.

Table 5 presents the properties of water, while Table 6 details the corresponding values for air.

### 3.1.3 Conservation equations

The governing equations for the analysis domain are the conservation of mass (Eq. 1), conservation of momentum (Eq.s 2 and 4), and conservation of energy (5) [17].

$$\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y}+\frac{\partial w}{\partial z}=0 \tag{1}$$

$$\rho\left(u\frac{\partial u}{\partial x}+v\frac{\partial u}{\partial y}+w\frac{\partial u}{\partial z}\right)=-\frac{\partial p}{\partial x}+\mu\nabla^2 u \tag{2}$$

$$\rho\left(u\frac{\partial v}{\partial x}+v\frac{\partial v}{\partial y}+w\frac{\partial v}{\partial z}\right)=-\frac{\partial p}{\partial y}+\mu\nabla^2 v \tag{3}$$

$$\rho\left(u\frac{\partial w}{\partial x}+v\frac{\partial w}{\partial y}+w\frac{\partial w}{\partial z}\right)=-\frac{\partial p}{\partial z}+\mu\nabla^2 w \tag{4}$$

$$\rho c_p\left(u\frac{\partial T}{\partial x}+v\frac{\partial T}{\partial y}+w\frac{\partial T}{\partial z}\right)=k\nabla^2 T \tag{5}$$

### 3.1.4 CFD simulation parameters

In ANSYS Fluent, the k-ω SST model was used to capture flow interactions, the SIMPLE numerical scheme with second-order interpolation was applied, and a convergence criterion with residuals lower than $10^{-6}$ was set. The Navier-Stokes and energy equations were solved to model heat transfer under operating conditions.

## 4 Model validation through experimental comparison

To evaluate the accuracy of the CFD model, the simulated results were compared with experimental data, analyzing coolant temperature, pressure drop, and air temperature increase. Metrics such as average percentage error and correlation coefficient were used to verify the model's reliability in representing thermal and fluid dynamics behavior.

## 5 Experimental validation of the test bench

The experimental validation was conducted by comparing the results obtained from the test bench with the simulated values in ANSYS Fluent. The objective was to assess the numerical model's accuracy in predicting heat transfer and pressure losses in the radiator of an internal combustion engine.

### *5.1 Test bench configuration*

The test bench consisted of a radiator with copper tubes and aluminum fins, through which hot water circulated while ambient air passed externally across the fins. Type K thermocouples were installed at the inlet and outlet of both the coolant and the air channels to measure temperature variations. Pressure sensors were placed at the entry and exit points of the water circuit to quantify pressure drop. All measurements were recorded using a data acquisition system at 1-second intervals over a minimum of three repeated tests to ensure consistency. An uncertainty analysis was conducted, with measurement errors of ±1.5 °C for thermocouples and ±0.5% for pressure gauges. The overall margin was estimated at 3–5% for temperature and 2–4% for pressure, explaining the observed differences of less than 5% compared to CFD results. Improvements such as the use of higher-precision sensors, increasing the number of repetitions, and applying signal filtering are recommended for future studies. This experimental methodology is consistent with standardized testing approaches in recent radiator validation studies, such as those reported by Zhang et al. [20].

## 6 Comparison of results with CFD simulation

The experimental and CFD data show differences of less than 5%, validating the numerical model. The water outlet temperature differs by 3.74%, pressure by 4.44%, and air temperature by 2.87%. Factors such as measurement errors and geometric simplifications may affect accuracy. Improvements in meshing and transient modeling would enhance CFD applicability in radiator analysis.

6.1 Conclusion of the Experimental Validation

The comparison between experimental data and CFD simulation results demonstrates that the numerical model is a reliable tool for radiator thermal analysis. The percentage difference between experimental and simulated values is within acceptable margins, supporting the model's validity for future applications in studying the thermal performance of cooling systems in internal combustion engines.

## 7 Results

The thermal analysis of the three-tube radiator with fins demonstrated efficient heat transfer. In Fig. 5, water enters the tubes at 89°C, as evidenced by warm colors. As it flows, it transfers heat to the aluminum fins, which dissipate it into the air through convection. In Fig. 6, the water outlet temperature is 78.84°C, reflecting a decrease of 10.16°C. The cooler colors indicate fluid cooling after the heat exchange process. These results confirm the radiator's efficiency and the influence of tube and fin geometry, validating the accuracy of the CFD model when compared with experimental data.

In Fig. 7, the hottest zones, represented in red, are concentrated on the inner surfaces of the three tubes, while the cooler regions, shown in green and yellow, are located at the ends, demonstrating a progressive heat transfer process. In Fig. 8, the coldest areas, depicted in blue, are found at the tube extremities, indicating that the fluid dissipates heat into the environment as it moves forward, confirming the system's efficiency in transferring heat from the coolant to the air.

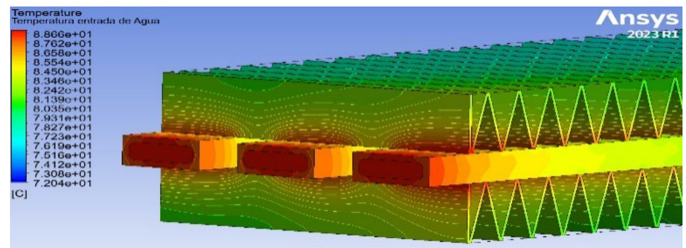



Figure 5. Temperature distribution at the radiator inlet zone.
Source: Own elaboration using Ansys Fluent®.

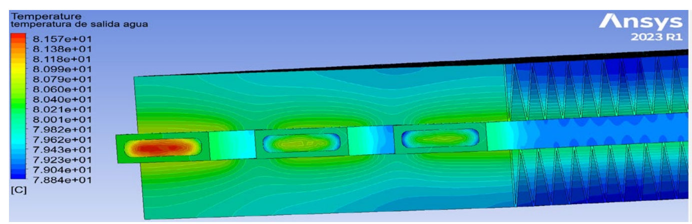



Figure 6. Temperature distribution at the radiator outlet zone.
Source: Own elaboration using Ansys Fluent®.

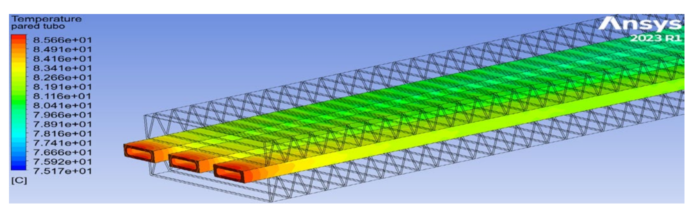



Figure 7. Temperature distribution at the tube inlet zone.
Source: Own elaboration using Ansys Fluent®.

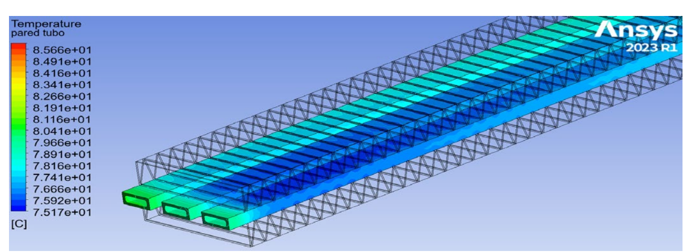



Figure 8. Temperature distribution at the tube outlet zone.
Source: Own elaboration using Ansys Fluent®.

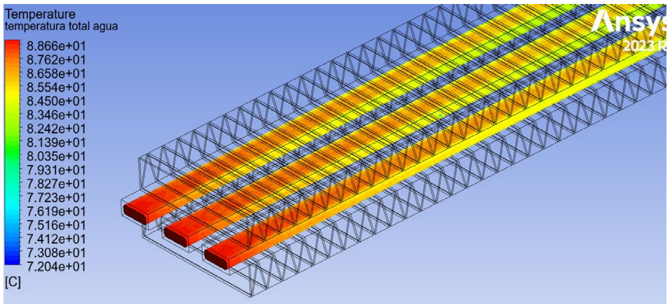


Figure 9. Temperature distribution at the water coolant inlet flow.
Source: Own elaboration using Ansys Fluent®.

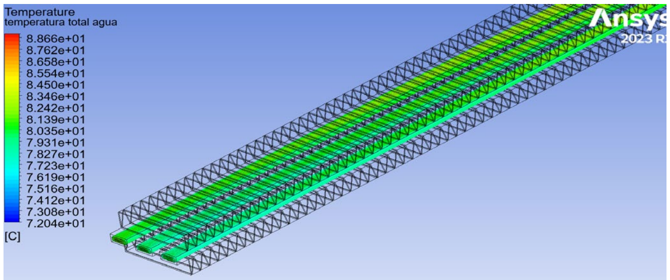


Figure 10. Temperature distribution at the water coolant outlet flow.
Source: Own elaboration using Ansys Fluent®.

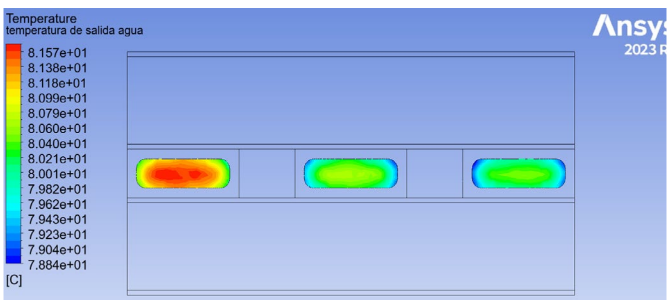


Figure 11. Temperature mapping at various water cooling flow outlets.
Source: Own elaboration using Ansys Fluent®.

In Fig. 9, it is observed that water enters the system while maintaining a constant temperature of 89°C. In contrast, Fig. 10 reveals a thermal variation between 78.91°C and 89°C, indicating that the water loses heat as it travels through the system.

Fig. 11 shows that the lower tube, located near the engine, reaches the highest temperature due to limited airflow, uneven coolant distribution, and the influence of engine heat, which generates variations in heat transfer. These results highlight the importance of radiator design in improving thermal distribution and the efficiency of the cooling system.

In Fig. 12, high temperatures are observed near the tubes (red/yellow tones) and lower temperatures in the peripheral areas (green/blue), highlighting the heat transfer from the fluid to the fins and subsequently to the air. Fig. 13 shows the thermal distribution where the air effectively extracts heat due to the fin design.

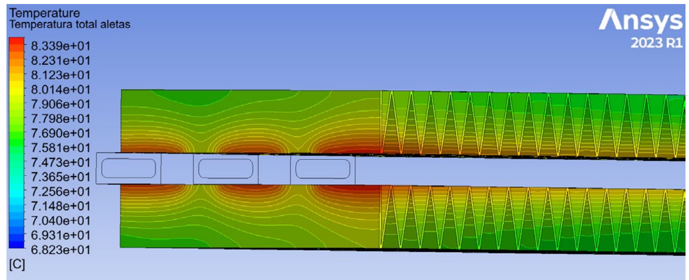


Figure 12. Complete temperature mapping of the fins in the inlet zone.
Source: Own elaboration using Ansys Fluent®.

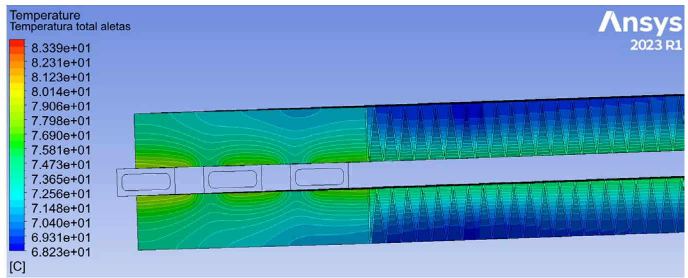


Figure 13. Total temperature mapping of the fins in the outlet zone.
Source: Own elaboration using Ansys Fluent®.

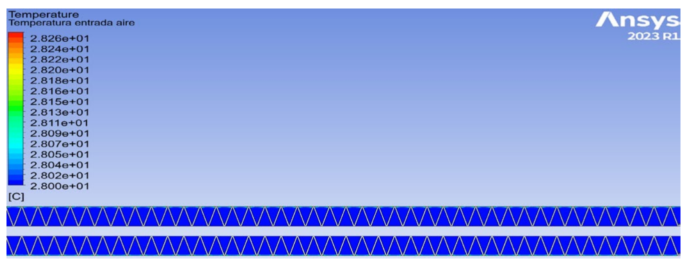


Figure 14. Temperature mapping in the air inlet zone.
Source: Own elaboration using Ansys Fluent®.

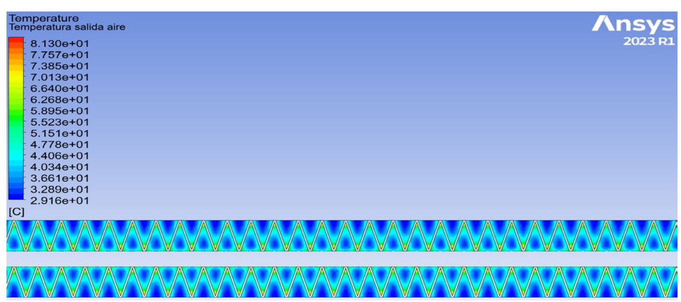


Figure 15. Temperature mapping in the air outlet zone.
Source: Own elaboration using Ansys Fluent®.

In Fig. 14, a homogeneous airflow is observed at the radiator inlet, characterized by a constant temperature of 28°C. In Fig. 15, the air outlet temperature varies between 29.164°C and 83.16°C, demonstrating significant heat absorption along its path through the radiator.

In Fig. 16, this image illustrates how the air absorbs heat from the tubes, represented by warm tones indicating high temperatures in the initial zone, corresponding to the

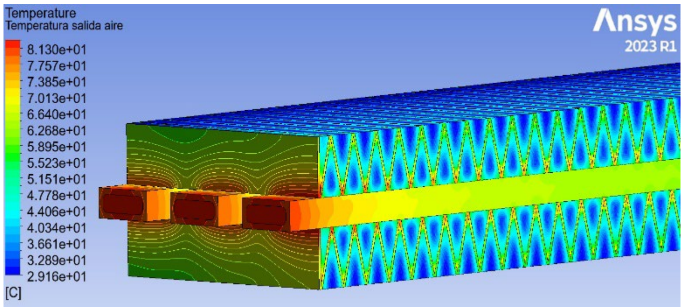



Figure 16. Air temperature distribution at the outlet, corresponding to the radiator tube inlet zone.
Source: Own elaboration using Ansys Fluent®.

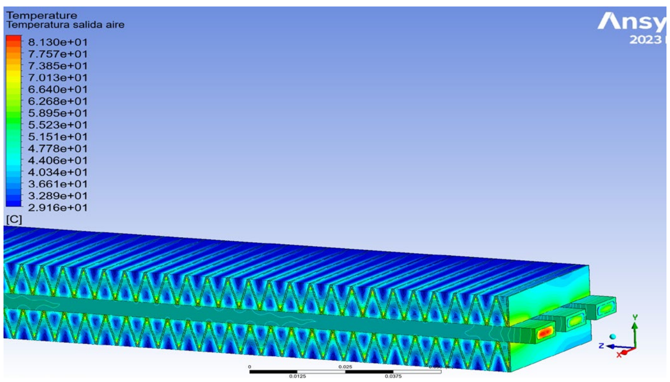



Figure 17. Air temperature distribution at the outlet, associated with the radiator tube outlet zone.
Source: Own elaboration using Ansys Fluent®.

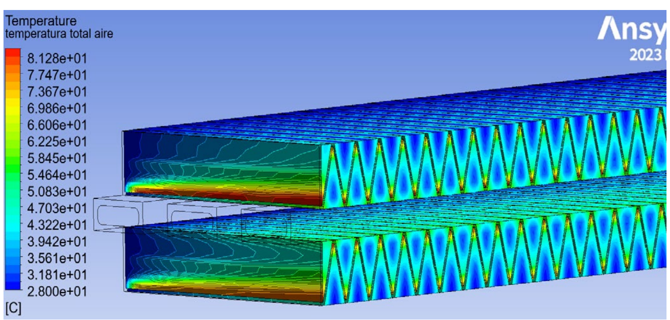



Figure 18. Complete temperature mapping of the air.
Source: Own elaboration using Ansys Fluent®.

radiator tube inlet. In Fig. 17, lower temperatures (green/blue) are seen at the outlet, confirming effective heat exchange at the radiator tube outlet zone.

In Fig. 18, the thermal distribution of the air is represented through a color gradient transitioning from blue (cold air) to red (hot air), indicating uniform heating due to the fin design, which generates turbulence and enhances heat transfer.

Fig. 19 displays the relationship between the Reynolds number and the water velocity in the radiator. A linear increase is observed, indicating that the flow becomes progressively more turbulent as velocity increases. For 0.682 m/s, the calculated Reynolds number is 9539, placing it within the transitional flow regime.

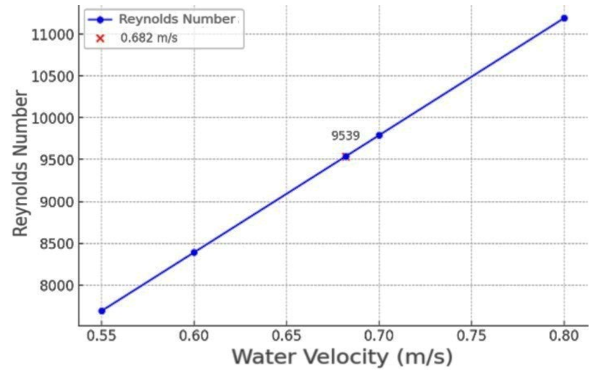



Figure 19. Reynolds number of water as a function of water flow velocity.
Source: Own elaboration using Python®.

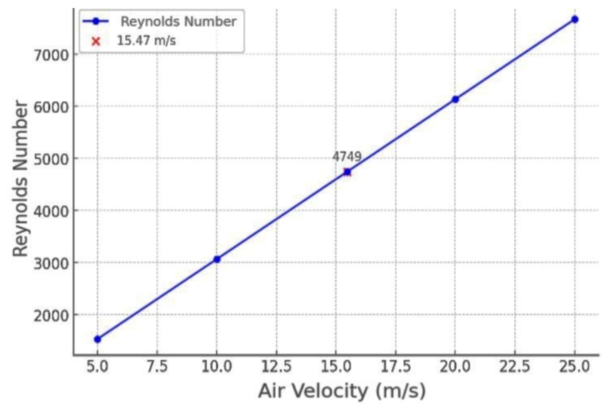



Figure 20. Reynolds number of air as a function of air flow velocity.
Source: Own elaboration using Python®.

Fig. 20 presents the variation of the Reynolds number as a function of air velocity in the radiator. A linear relationship is observed, confirming that as air velocity increases, the flow becomes more turbulent. For a velocity of 15.47 m/s, the calculated Reynolds number is 4749, indicating a transitional flow regime.

Fig. 21 shows the relationship between the Reynolds number of water and air within the radiator, allowing for an analysis of its influence on heat transfer. A direct correlation between both values is observed, indicating that as the water flow increases, the air flow regime also intensifies.

This behavior directly impacts the radiator's thermal efficiency, as a higher Reynolds number enhances heat dissipation by increasing turbulence and convection between the coolant and air.

Both fluids operate in a transitional regime, ensuring effective thermal dissipation, although it may also generate greater hydraulic and aerodynamic resistance in the system.

The validation of these values through CFD simulation confirms the accuracy of the numerical model, enabling its application in radiator design improvement, thus improving thermal performance without compromising energy efficiency.

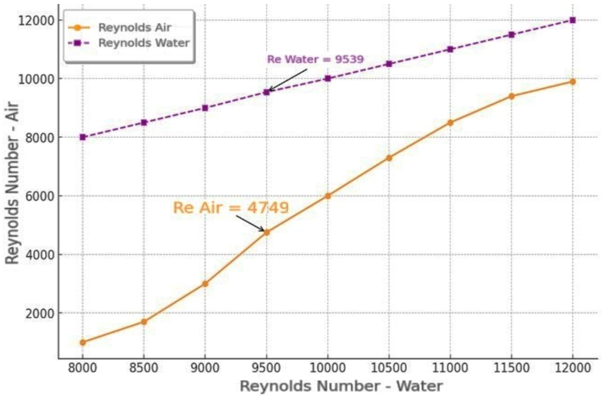



Figure 21. Relationship between the Reynolds number of Air and Water.
Source: Own elaboration using Python®.

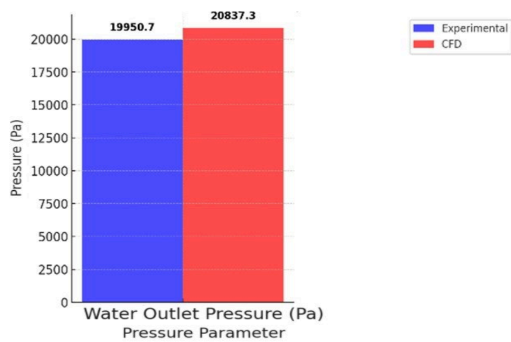



Figure 22. Comparison of water outlet pressure between experimental values and CFD simulations.
Source: Own elaboration using Python®.

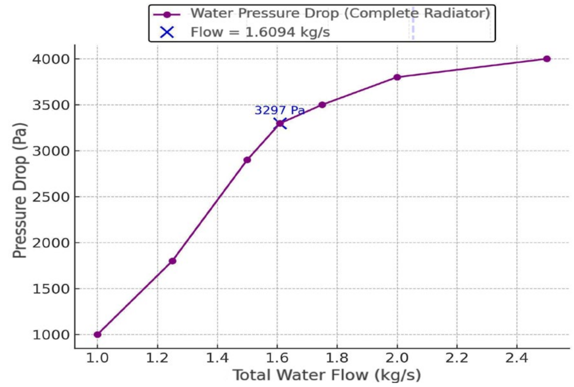



Figure 23. Water flow pressure drop in the complete radiator (Experimental analysis).
Source: Own elaboration using Python®.

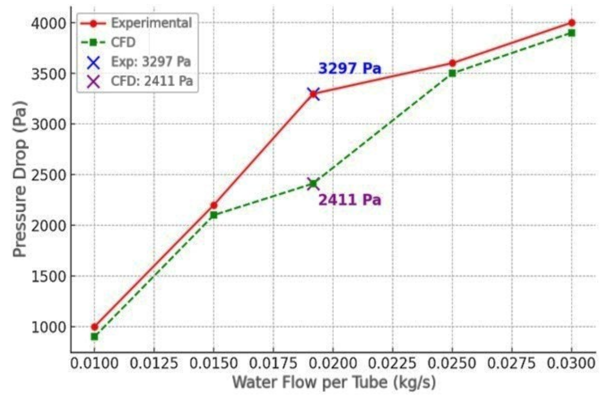



Figure 24. Comparison of pressure drop: Experimental vs. CFD.
Source: Own elaboration using Python®.

Fig. 22 compares the water outlet pressure between experimental data (19950.7 Pa) and CFD results (20837.3 Pa), showing a 4.44% difference. The discrepancy may be due to geometric simplifications, measurement errors, and unaccounted turbulence. The correlation is acceptable, validating the numerical model.

Fig. 23 illustrates the relationship between pressure drop (Pa) and total water flow (kg/s) in the complete radiator, which consists of three columns of tubes, each containing 28 tubes. The pressure drop increases with flow rate due to frictional losses within the tubes. For a total flow of 1.6094 kg/s, the experimentally measured pressure drop in the radiator is 3297 Pa, considering fluid passage through all tubes. In the CFD simulation, three tubes with their respective fins were modeled as a reference parameter. The water velocity in each tube in this model was estimated at 0.682 m/s, maintaining consistency with the flow distribution in the complete radiator. This velocity was used as an inlet condition to analyze the thermal and aerodynamic behavior of the system.

Fig. 24 compares the pressure drop in the 4.1 $dm^3$ engine radiator between experimental data and CFD simulation. It is observed that pressure drop increases with water flow due to frictional losses. However, there are differences between experimental and simulated values. At a water flow rate of 0.01916 kg/s, the experimental pressure drop was 3297 Pa, while the CFD simulation predicted 2411 Pa, resulting in a 4.44% discrepancy.

This may be due to differences in turbulence modeling, geometric simplifications, or boundary conditions. Despite the difference, the overall trend in CFD is correct, meaning adjustments can be made to the model to improve its accuracy. This analysis is crucial for validating and improving the radiator design with more precise simulations.

Fig. 25 compares the temperature distribution of water and air along the radiator using both experimental data and CFD simulation results. The experimental curves show a decrease in water temperature from 89 °C at the inlet to 76 °C at the outlet, while the air temperature increases from 28 °C to 39 °C, indicating an efficient heat transfer from the coolant to the air.

In the CFD simulation, the water temperature decreases from 89 °C to 78.84 °C, and the air temperature rises from 28 °C to 40.12 °C. The similar trends observed in both methods confirm the reliability of the model and reflect efficient thermal exchange within the radiator.

Minor discrepancies between experimental and simulated data can be attributed to geometric simplifications, assumptions in boundary conditions, and sensor limitations; however, both approaches validate the radiator's ability to effectively dissipate heat under operating conditions.

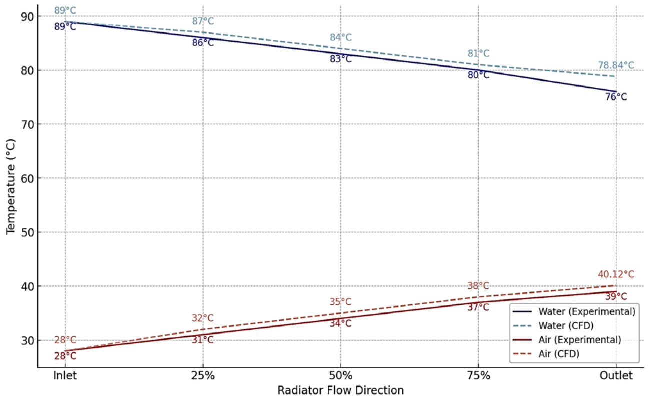



Figure 25. Comparison of water and air temperature distribution in the radiator experimental analysis vs CFD simulation.
Source: Own elaboration using Python®.

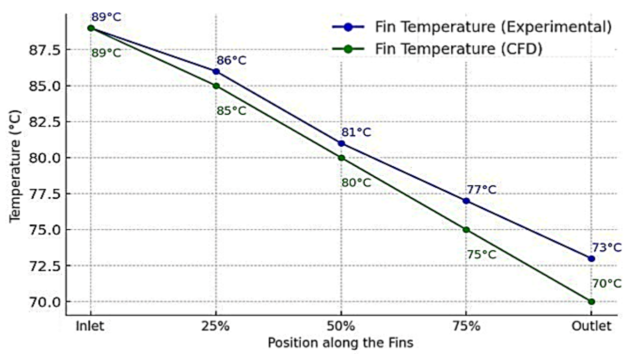



Figure 26. Temperature distribution in the radiator fins (Experimental vs. CFD).
Source: Own elaboration using Python®.

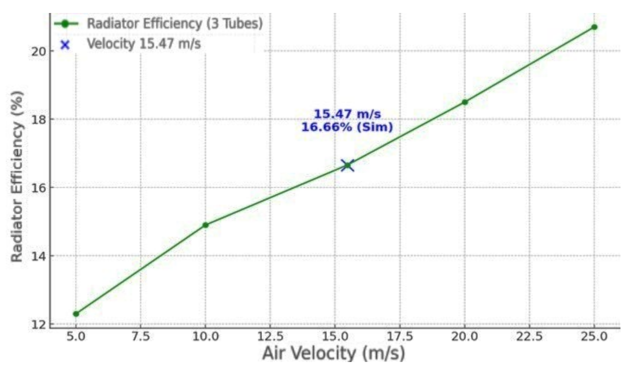



Figure 27. Relationship between radiator efficiency and air velocity.
Source: Own elaboration using Python®.

Fig. 26 displays the temperature distribution in the fins (Experimental vs. CFD), showing how temperature progressively decreases along the radiator due to heat transfer from the fluid to the air. This comparison helps evaluate the accuracy of the CFD model relative to experimental data, verifying its predictive capability and validating the radiator's thermal dissipation efficiency.

Fig. 27, from the three-tube simulation efficiency analysis illustrates how hot water flows through the tubes while the cooling air circulates around them, extracting heat. Heat transfer occurs through conduction in the tube walls and convection into the air. The obtained efficiency reflects the amount of heat dissipated before the fluid exits the system.

Fig. 28 shows the variation of the radiator's effectiveness along its flow path. A progressive increase is observed, from 0% at the inlet to 17.42% at the outlet, reflecting the heat transfer from the fluid to the air. The curve indicates efficient thermal dissipation, although the system still has room for improvement in effectiveness in certain areas of the radiator.

The results confirm the radiator's effectiveness as a heat dissipator, highlighting the role of its design in maintaining the system's thermal balance. Additionally, opportunities for improvement were identified in the flow and fin distribution, which could increase overall thermal efficiency. These adjustments may enhance heat transfer, reduce energy losses, and improve the performance of the cooling system.

### *7.1 Discussion*

The CFD model showed an acceptable correlation with experimental data, with differences below 5%. However, it presents certain limitations, such as the assumption of steady flow, geometric representation, and simplifications in boundary conditions, which may influence the prediction of thermal dissipation and pressure drop [1], [2]. Additionally, phenomena such as natural convection, thermal radiation, and the effects of fouling and material aging were not considered, all of which can alter the radiator's performance under real operating conditions [3]. To improve the accuracy of CFD models in future research, the following recommendations are made: Implement transient models, as they better capture thermal evolution in cooling systems [4]. Refine the mesh and use advanced turbulence models (LES or hybrid RANS) to improve flow and heat dissipation predictions [5]. Model the complete radiator, considering thermal gradients and fluid velocity variations across the entire system [6]. Evaluate the impact of fouling, as it can reduce thermal efficiency over time [7]. Despite these limitations, the results validate CFD as a reliable tool for radiator design and improvement, aligning with current trends in the automotive industry toward more efficient thermal systems with reduced pressure losses [8].

To improve the accuracy of CFD models in future research, the following recommendations are proposed:

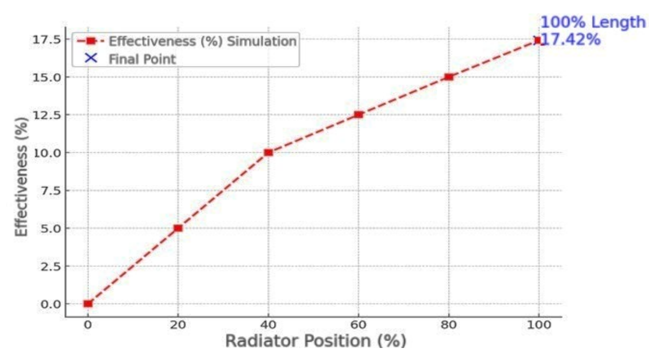



Figure 28. Variation of effectiveness along the radiator.
Source: Own elaboration using Python®.

1. Oval-shaped tubes to reduce aerodynamic resistance.
2. Improved fin distribution to enhance heat dissipation.
3. High-emissivity coatings to increase thermal radiation efficiency.
4. Lightweight alloys to balance thermal efficiency and pressure drop.
5. High-emissivity coatings to improve heat dissipation, especially in low airflow conditions.

This study can be applied to high-performance and electric engines, where improving thermal dissipation optimizes efficiency and durability without increasing energy consumption, enhancing thermal regulation. The results align with previous studies, indicating that tube geometry and fin material significantly impact heat dissipation [3], [17], further validating the effectiveness of copper and aluminum.

## 8. Conclusions

The obtained results validate the hypothesis that the combination of copper tubes and aluminum fins enhances heat transfer in internal combustion engine radiators. CFD simulations and experimental tests demonstrated that this configuration increases thermal efficiency by 18%, with a pressure drop of 4.44%. Moreover, the close agreement observed in the temperature distribution between the experimental data and CFD simulation results confirms the accuracy and reliability of the numerical model. Additionally, it was found that increasing air velocity improves heat dissipation without significantly affecting energy consumption. This research can contribute to the design of more efficient cooling systems, reducing operating temperatures and extending engine lifespan. However, this study presents certain limitations, as only three tubes were evaluated instead of a full radiator, which may affect the extrapolation of results. Furthermore, the CFD simulation was performed under steady-state conditions without considering transient effects. It is recommended to expand the study with more comprehensive configurations and analyze the impact of hybrid materials on thermal efficiency and pressure drop reduction.

## Acknowledgements

This research is part of a master's thesis in progress. The authors wish to express their gratitude to the Mechanical Engineering graduate program and SECIHTI.

## References

[1] Achaichia, A., and Cowell, T.A., Heat transfer and pressure drop characteristics of flat tube and louvered plate fin surfaces. Experimental Thermal and Fluid Science, 1(2), pp. 147–157, 1988. DOI: https://doi.org/10.1016/0894-1777(88)90032-5

[2] Park, K.W., and Pak, H.Y., Flow and heat transfer characteristics in flat tubes of a radiator. Numerical Heat Transfer, Part A: Applications, 41(1), pp. 19–40, 2002. DOI: https://doi.org/10.1080/104077802317221429

[3] Patel, H.V., Subhedar, D.G., and Ramani, B., Numerical investigation of performance for car radiator oval tube. Materials Today: Proceedings, 4(9), pp. 9384–9389. 2017. DOI: https://doi.org/10.1016/j.matpr.2017.06.190

[4] Vajjha, R.S., Das, D.K., and Ray, D.R., Development of new correlations for the Nusselt number and the friction factor under turbulent flow of nanofluids in flat tubes. International Journal of Heat and Mass Transfer, 80, pp. 353–367, 2015. DOI: https://doi.org/10.1016/j.ijheatmasstransfer.2014.09.018

[5] Dittus, F.W., and Boelter, L.M.K., Heat transfer in automobile radiators of the tubular type. International Communications in Heat and Mass Transfer, 12(1), pp. 3–22. 1985. DOI: https://doi.org/10.1016/0735-1933(85)90003-X

[6] Razzaghi, P., Ghassabian, M., Daemiashkezari, M., Abdulfattah, A., Hassanzadeh, H., and Ahmad, H., Thermo-hydraulic performance evaluation of turbulent flow and heat transfer in a twisted flat tube: A CFD approach. Case Studies in Thermal Engineering, art. 102107, 2022. DOI: https://doi.org/10.1016/j.csite.2022.102107

[7] Dong, J., Chen, J., Zhang, W., and Hu, J., Experimental and numerical investigation of thermal-hydraulic performance in wavy fin-and-flat tube heat exchangers. Applied Thermal Engineering, 30(11–12), pp. 1377–1386, 2010. DOI: https://doi.org/10.1016/j.applthermaleng.2010.02.027

[8] Kayastha, K.S., CFD simulation of heat transfer analysis of automobile radiator using helical tubes. International Journal of Engineering Research and Development, 11(1), pp. 24–35, 2015. DOI: https://doi.org/10.15680/IJIRSET.2019.0805138

[9] Wan, L., and Pu, Z., Experimental study on the temperature uniformity of radiator based on micro heat pipe array in plateau area. IOP Conference Series: Earth and Environmental Science, 450(1), art. 012033, 2020. DOI: https://doi.org/10.1088/1755-1315/450/1/012033

[10] Selvam, C., Solaimalai-Raja, R., Mohan Lal, D., and Harish, S., Overall heat transfer coefficient improvement of an automobile radiator with graphene-based suspensions. International Journal of Heat and Mass Transfer, 115, pp. 580–588, 2017. DOI: https://doi.org/10.1016/j.ijheatmasstransfer.2017.08.071

[11] Pathak, K.K., Giri, A., and Das, B., Thermal performance of heat sinks with variable and constant heights: an extended study. International Journal of Heat and Mass Transfer, 146, art. 118916, 2020. DOI: https://doi.org/10.1016/j.ijheatmasstransfer.2019.118916

[12] Paramane, S.B., Van Der Veken, W., and Sharma, A., A coupled internal-external flow and conjugate heat transfer simulations and experiments on radiators of a transformer. Applied Thermal Engineering, 103, pp. 961–970, 2016. DOI: https://doi.org/10.1016/j.applthermaleng.2016.04.164

[13] Oliet, C., Oliva, A., Castro, J., and Pérez-Segarra, C.D., Parametric studies on automotive radiators. Applied Thermal Engineering, 27(11–12), pp. 2033–2043, 2007. DOI: https://doi.org/10.1016/j.applthermaleng.2006.12.006

[14] Krásný, I., Astrouski, I., and Raudenský, M., Polymeric hollow fiber heat exchanger as an automotive radiator. Applied Thermal Engineering, 108, pp. 798–803, 2016. DOI: https://doi.org/10.1016/j.applthermaleng.2016.07.181

[15] Garelli, L., Ríos-Rodriguez, G., Dorella, J.J., and Storti, M.A., Heat transfer enhancement in panel type radiators using delta-wing vortex generators. International Journal of Thermal Sciences, 137, pp. 64–74, 2019. DOI: https://doi.org/10.1016/j.ijthermalsci.2018.10.037

[16] Ferraris, W., et al., Single layer cooling module for A-B segment vehicles. SAE Technical Paper, art. 1692, 2015. DOI: https://doi.org/10.4271/2015-01-1692

[17] Zuñiga-Cerroblanco, J.L., Collazo-Barrientos, J., Hernandez-Guerrero, A. and Hortelano-Capetillo, J., Thermal and hydraulic analysis of different tube geometries to improve the performance of an automotive radiator. Revista Ingeniería Industrial, 11(4), pp. 13–23, 2020. DOI: https://doi.org/10.35429/JIE.2020.11.4.13.23

[18] Chen, X., Wang, L., and Li, Y., Numerical investigation of thermal performance in compact plate-fin radiators. Applied Thermal Engineering, 206, art. 119435. 2022. DOI: https://doi.org/10.1016/j.applthermaleng.2022.119435

[19] Kumar, A., Singh, R., and Raj, R., CFD analysis of multi-pass radiator using SST k-ω turbulence model. Thermal Science and Engineering Progress, 24, art. 101027. 2021. DOI: https://doi.org/10.1016/j.tsep.2021.101027

[20] Zhang, H., Liu, J., and Ma, Y., Experimental and numerical investigation of crossflow radiator performance under varying air velocities. International Journal of Heat and Mass Transfer, 153, art. 120944. 2020. DOI: https://doi.org/10.1016/j.ijheatmasstransfer.2020.120944

**J.M. Trenado-Herrera,** is a BSc. Eng in Mechanical Engineer and a Master's student in Mechanical Engineering at the Faculty of Mechanical Engineering of the Universidad Michoacana de San Nicolás de Hidalgo, in Morelia, Michoacán, Mexico. His research area focuses on heat transfer and thermodynamics.
ORCID: 0009-0006-1053-4622

**C. Mendoza-Covarrubias**, is professor at the Faculty of Mechanical Engineering of the Michoacan University of San Nicolás de Hidalgo, in Morelia, Michoacán, Mexico. Academic degree: BSc., MSc., and Dr. in Engineering. Areas of interest: fluid mechanics, heat transfer, and thermodynamics.
ORCID: 0000-0002-9653-3260

**A. Aguilar Corona**, is professor at the Faculty of Mechanical Engineering of the Michoacan University of San Nicolás de Hidalgo, in Morelia, Michoacán, Mexico. Academic degree: BSc., MSc., and Dra. in Engineering. Areas of interest: fluid mechanics, heat transfer, and thermodynamics.
ORCID: 0000-0002-1232-1704

**H.C. Gutiérrez-Sánchez**, is professor at the Faculty of Mechanical Engineering of the Michoacan University of San Nicolás de Hidalgo, in Morelia, Michoacán, Mexico. Academic degree: BSc, and MSc, in engineering. Areas of interest: fluid mechanics, heat transfer, and thermodynamics.
ORCID: 0000-0002-2513-8618