Thermal Analysis Of A Compact Exchanger With The Aid Of Computational Fluid Dynamics

Author

[1] Vinicius De Oliveira Menezes Sirqueira, Laboratory for Applied Mechanical Design and Technology (LAPMET), School of Mechanical Engineering, State University of Santa Cruz (UESC), Campus Soane Nazaré de Andrade, Rodovia Jorge Amado, km 16, CEP: 45662-900, Ilhéus, Bahia, Brazil.

[2] Ian De Souza Santos, Laboratory for Applied Mechanical Design and Technology (LAPMET), School of Mechanical Engineering, State University of Santa Cruz (UESC), Campus Soane Nazaré de Andrade, Rodovia Jorge Amado, km 16, CEP: 45662-900, Ilhéus, Bahia, Brazil.

[3] Jhonatan Silva Farias, Laboratory for Applied Mechanical Design and Technology (LAPMET), School of Mechanical Engineering, State University of Santa Cruz (UESC), Campus Soane Nazaré de Andrade, Rodovia Jorge Amado, km 16, CEP: 45662-900, Ilhéus, Bahia, Brazil.

[4] Felippe Lopes Passarinho, Laboratory for Applied Mechanical Design and Technology (LAPMET), School of Mechanical Engineering, State University of Santa Cruz (UESC), Campus Soane Nazaré de Andrade, Rodovia Jorge Amado, km 16, CEP: 45662-900, Ilhéus, Bahia, Brazil.

[6] André Luis Vinagre Pereira, Laboratory for Applied Mechanical Design and Technology (LAPMET), School of Mechanical Engineering, State University of Santa Cruz (UESC), Campus Soane Nazaré de Andrade, Rodovia Jorge Amado, km 16, CEP: 45662-900, Ilhéus, Bahia, Brazil.

[7] Erickson Fabiano Moura Sousa Silva, Laboratory for Applied Mechanical Design and Technology (LAPMET), School of Mechanical Engineering, State University of Santa Cruz (UESC), Campus Soane Nazaré de Andrade, Rodovia Jorge Amado, km 16, CEP: 45662-900, Ilhéus, Bahia, Brazil.

Corresponding Author: [1]Vinicius De Oliveira Menezes Sirqueira, Laboratory for Applied Mechanical Design and Technology (LAPMET), School of Mechanical Engineering, State University of Santa Cruz (UESC), Campus Soane Nazaré de Andrade, Rodovia Jorge Amado, km 16, CEP: 45662-900, Ilhéus, Bahia, Brazil.

Abstract: *In various equipment present in our daily lives, whether in domestic or industrial environments, it is of paramount importance that efficient heat exchanges occur between the working fluid (WF) and the refrigerant so that the thermal cycle concludes and we achieve the expected result of this equipment. In this context, heat exchangers are widely used, since they employ mechanisms of exchange through convection and conduction, without the need for direct contact between fluids, thus allowing the removal of heat from the WF. Among the various types of heat exchangers, compact ones stand out due to their uniqueness in providing a high rate of heat transfer in a small volume. This makes them ideal for situations that demand less weight and size. With that said, the aim of this work was to analyze the Computational Fluid Dynamics (CFD) process in a model of a compact heat exchanger. Different geometric and operational parameters present in the composition and functioning of this process were sought to be verified in order to achieve its validation. To achieve this goal, evaluations of different configurations of compact heat exchangers based on bibliographic studies were carried out. With the definition of the analysis equipment, the computational model was developed using Computer-Aided Design (CAD) with SolidWorks software, and fluid dynamic simulations were performed using Flow Simulation. The results of the modeling of the compact heat exchanger showed a mesh validated analytically with the calculation of the Reynolds number due to deviations of 0.11% at a temperature of 50°C and 0.81% at a temperature of 250°C, and experimentally with the results developed by Ma et al. (2020) observing a maximum variation of 13.4% in the cold outlet temperature in case 1 and a minimum deviation of 9.75% in the hot outlet in case 3, demonstrating effectiveness in the validation process. Thus, it is evidenced that with the developed CFD module, it is possible to analyze different configurations of compact exchangers.*

Keywords: *Compact heat exchanger, Computacional Fluid Dynamics (CFD), Heat transfer.*

I. INTRODUCTION

Heat exchangers are used in both industrial sectors and everyday life, on small and large scales. Their specifications are defined according to the application needs and operating environment. These devices, for example, allow for more comfortable temperatures in living spaces and enable the rapid and efficient cooling of large machines in the industrial sector (Incropera et al., 2014). Among the various models available, the compact heat exchanger is the most commonly found in everyday life. As the name suggests, it is a small model found in devices such as car radiators and air conditioners. This model, in particular, aims to provide the highest possible thermal exchange on small heat transfer surfaces (Yousefi et al, 2015).

Compact heat exchangers feature a cross-flow system, meaning the fluids cross paths inside the heat exchanger, flowing perpendicularly to each other. These models are used when a large heat transfer surface area per unit volume is needed. This ratio is known as the area density, β. A compact heat exchanger is characterized by having $\beta > 700$ m²/m³. Therefore, they are suggested for applications where there are weight and space constraints. An example of a compact heat exchanger is the radiator in vehicles and water coolers for computer cooling (Çengel, 2012).

In the tube and fin configuration, the compact heat exchanger is constructed with plates that act as fins. These plates are mounted on a set of tubes distributed throughout the plate. Heat exchange in this device occurs through the walls of the tubes and fins. The operational process involves air flowing between the fins while the refrigerant fluid flows inside the tubes (MA, Xiaowei et al, 2020).

To design and use a heat exchanger for a specific application, it is necessary to consider the main information about its operating conditions and performance. The heat transfer process in a heat exchanger usually occurs between two fluids. According to the second law of thermodynamics, heat is transferred from the hot fluid to the wall by convection, then through the wall by conduction, and finally from the wall to the cold fluid by convection again. The radiation mechanism is accounted for within the convection coefficient (ÇENGEL; GHAJAR, 2012).

Knowing the required thermal exchange demand in the operation, the overall heat transfer coefficient is of utmost importance. With the coefficient determined, it is possible to confirm that the heat exchanger is capable of performing the heat exchange process with the desired efficiency. Analyzing the overall heat transfer coefficient in an already functioning piece of equipment is a tool that enables the improvement and maintenance of the system according to operational needs (BARBOSA et al., 2017).

One of the most widely applied methods in the analysis of heat exchangers is the detailed study of the temperatures of the hot and cold fluids throughout the equipment. This analysis allows for an understanding of the thermal behavior of the exchanger and helps identify potential points for optimization (Incropera et al., 2014).

In this way, it is observed that the temperature distribution in a heat exchanger behaves such that the hot fluid entering the exchanger is cooled exponentially along the length of the exchanger until it exits. Conversely, the cold fluid is heated exponentially along the heat exchanger until it exits. Therefore, the appropriate way to use the average temperature distribution for studies in heat exchangers is in logarithmic form, as presented in Equation 1 below (ÇENGEL; GHAJAR, 2012):

$$\Delta T_{ml} = \frac{\Delta T_2 - \Delta T_1}{\ln (\Delta T_2 / \Delta T_1)} \tag{1}$$

Where ΔT_{ml} represents the Logarithmic Mean Temperature Difference (LMTD), and ΔT_1 and ΔT_2 are the temperature differences between the two fluids at the inlet and outlet of the heat exchanger. The temperature difference within the operational process is determined by the equipment's design and the fluid flow direction.

The LMTD (Logarithmic Mean Temperature Difference) is used when analyzing the thermal exchange behavior throughout the heat exchanger. This allows for the efficient design or prediction of the equipment's performance for different applications. The application of LMTD is crucial when there is a need to control the rate of cooling or heating of the fluids and in the pursuit of improving thermal exchange efficiency (Incropera et al., 2014).

The Reynolds number is a dimensionless quantity widely used in fluid mechanics to characterize the behavior of a flow. It is defined as the ratio of inertial forces to viscous forces in a moving fluid. The Reynolds number is calculated by multiplying the fluid velocity by the characteristic dimension of the flow and dividing this value by the fluid's viscosity, as presented in Equation 2 below (Çengel & Ghajar, 2012).

$$Re = \frac{\rho \cdot \upsilon \cdot D}{\mu} \tag{2}$$

Where ρ is the fluid density (kg/m³), υ is the fluid velocity (m/s), and D is the diameter of the tube (m).

The Reynolds number is crucial in the design and operation of heat exchangers. By determining whether the flow inside the tubes is laminar or turbulent, the Reynolds number directly influences heat transfer and pressure drop in the exchangers (Çengel & Ghajar, 2012).

The methods used for continuous improvement of these devices involve the use of mathematical and computational tools; the numerical method of Computational Fluid Dynamics (CFD) is one of the most efficient tools in the analysis and optimization of equipment in the thermo-fluid area (Rodrigues, 2019). Before the advent of current numerical simulation technologies, optimizations required significant time and numerous test specimens for experimental analysis. The use of CFD in the optimization of equipment like heat exchangers has proven to be a cost-effective, sustainable, and efficient complement in the quest for equipment improvement (Ma et al., 2020).

Thus, this work aimed to analyze the process of Computational Fluid Dynamics (CFD) in a compact heat exchanger model. The goal was to investigate different geometric and operational parameters present in the composition and operation of this process to achieve its validation. In this way, it promotes the use of CFD tools for optimization and problem-solving in the industry.

II. EXPERIMENTAL PROCEDURE

The first step of this work consists of a literature review, where the results of the work developed by Ma et al. (2020) were studied. The focus will be on the experimental analysis of the outlet air temperature in a compact heat exchanger under specific operational conditions. Next, the heat exchanger model was created using Computer-Aided Design (CAD) and the CFD model was configured in Flow Simulation. Initially, the model was validated with analytical results under standard conditions and subsequently with the same conditions as the experiment. Experiments with different mesh configurations were conducted to obtain results that were more accurate and closer to the reference experiment. The comparison between experimental and simulated results allowed for the validation of the CFD model, paving the way for other computational analyses aimed at optimizing the heat exchanger.

The geometric model of the heat exchanger sketched for validation follows the parameters of the experimental model. Thus, the heat exchanger configuration consists of 24 tubes arranged in a Z-shape, with the working fluid being the R22 refrigerant in the experimental validation, as illustrated in Fig. 2 and detailed in Table 1.

Boundary conditions were applied to the CFD model to validate the numerical computational model with the conditions of the experiments conducted by Ma et al. (2020). The boundary conditions applied to the numerical model, following the experimental conditions, are presented in Table 2 below:

Figure 2. Sketched compact heat exchanger.

Table 1. Measurements and configurations of the model

Parameter	Measurement
Tube Length (mm)	410
Heat Exchanger Height (mm)	305
Heat Exchanger Depth (mm)	38,1
Longitudinal Tube Pitch (mm)	25,4
Transverse Tube Pitch (mm)	19,5
Outer Tube Diameter (mm)	9,53
Tube Wall Thickness (mm)	0,3
Fin Thickness (mm)	0,14
Fin Pitch (mm)	1,6
Number of Tube Columns	2
Number of Tubes per Column	12
Number of Fins	60
Number of Fins per Meter	630

Table 2. Boundary conditions of the experimental model.

*	T_{rin}	P_{in}	M_r	P_{out}	V_a	T_{ain}	T_{rout}	T_{aout}
Case 1	88,1	1,69	52	1,66	1,2	22,1	45,6	40,7
Case 2	89,9	1,66	60	1,6	1,68	21	45	38,5
Case 3	88,5	1,65	61	1,58	1,68	21,1	44,2	38,2
Case 4	90,2	1,67	62	1,59	1,68	21,1	44,7	38,2

The following parameters are defined: $Trin$ is the inlet temperature on the refrigerant side (°C), Pin is the inlet pressure on the refrigerant side (MPa), Mr is the mass flow rate on the refrigerant side (kg/h), $Pout$ is the outlet pressure on the refrigerant side (MPa), and Va is the inlet air velocity (m/s). $Tain$ is the inlet temperature on the air side (°C), $Trout$ is the outlet temperature on the refrigerant side (°C), and $Taout$ is the outlet temperature on the air side (°C).

Subsequently, the simulated results were compared with the analytical results using the Reynolds Equation, and then with the experimental results for each of the 4 cases proposed for the validation of the developed CFD model. The aim was to validate the efficiency of the model for heat exchanger analyses.

III. RESULTS AND DISCUSSIONS

The results obtained in this study for the validation of the mesh and CFD system for the configurations of the compact heat exchangers will be presented next.

3.1. Process for Mesh and CFD System Validation

The model validation process is meticulously carried out in several stages, with the aim of gradually adding complexity to the process and comparing the results obtained. This approach aims to achieve a mesh that produces results consistent not only with the analytical values from Equation 6 but also with the experimental data by comparing the results from Ma et al. (2020). Through this careful validation, we seek to ensure the reliability and accuracy of the model. The results obtained in this validation process will be presented in detail below.

III.1.1 Validation of a Single Tube

The first stage of the CFD model validation was carried out using only one tube from the heat exchanger, as shown in Fig. 3, to analyze the input and output parameters, comparing the numerical results with the analytical results

Figure 3. Illustration of the heat exchanger tube.
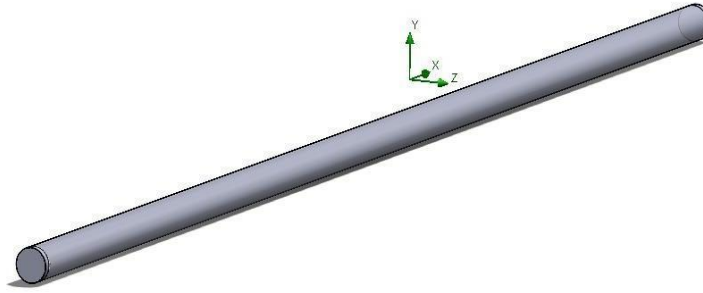
With the model defined, the numerical simulation of water flow in the heated tube was executed, considering the specified boundary conditions. The focus of the analysis was on temperature and Reynolds number, as illustrated in Figure 4.

Figure 4. Fluid heating simulation in a single tube of the heat exchanger.
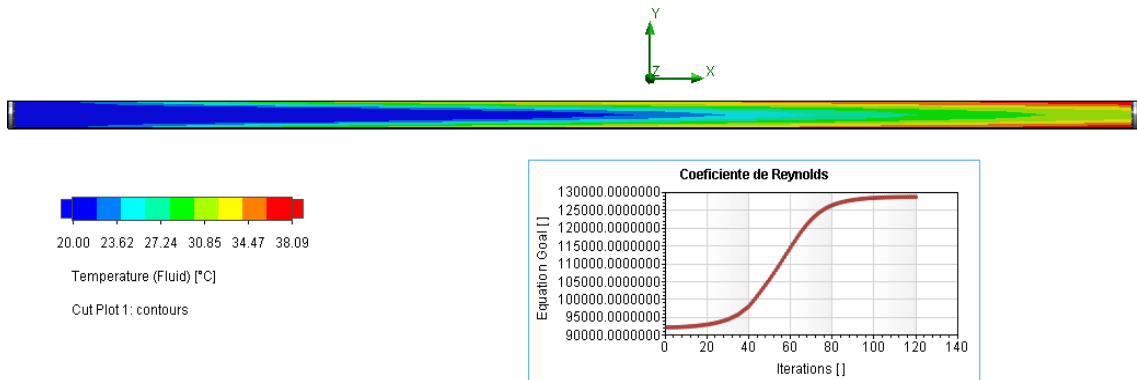
The next step in the model validation process is to compare the analytical and numerical results by varying the tube temperature, as illustrated in Table 3. The selected parameter for comparison is the Reynolds number, as there are variables directly dependent on the fluid temperature.

Table 3. Analytical and simulated values for a single tube.

| Pipe Temperature (°C)	Fluid Outlet Temperature (°C)	Reynolds Number (Analytical)	Reynolds Number (Simulated)	Variation (%)
50	24,83	103193,35	103217,97	0,03
100	35,26	128454,80	128604,13	0,12
150	47,55	160406,41	160898,14	0,31
200	60,94	197595,68	198516,34	0,47
250	74,99	238761,66	240138,23	0,58

The validation process was conducted using a single tube from the heat exchanger, where Reynolds numbers at different tube temperatures were compared, maintaining a constant flow velocity in all cases. The results revealed a deviation of only 0.03% at a temperature of 50°C and 0.68% at a temperature of 250°C. These results provided a preliminary validation of the proposed model, demonstrating its ability to accurately predict fluid behavior under different temperature conditions. This validation is a significant step towards the reliability and applicability of the model.

III.1.2 Validation of the Coil

In a subsequent stage of the CFD model validation, a more comprehensive and complex analysis of the system was conducted. The scope of the study was expanded to include the entire coil structure, accounting for its curves and broader fluid trajectories. This expansion allowed for a more precise and detailed analysis of the heat exchanger's behavior, considering all aspects of its design.

This expansion is clearly represented in Figure 5 below. In this figure, you can observe the inclusion of all the curves and trajectories of the coil, providing a more realistic and complete representation of the system. This more comprehensive analysis contributes to a more robust validation of the CFD model, providing consistent and reliable results for the thermal efficiency of the heat exchanger.

Figure 5. Illustration of the heat exchanger coil.
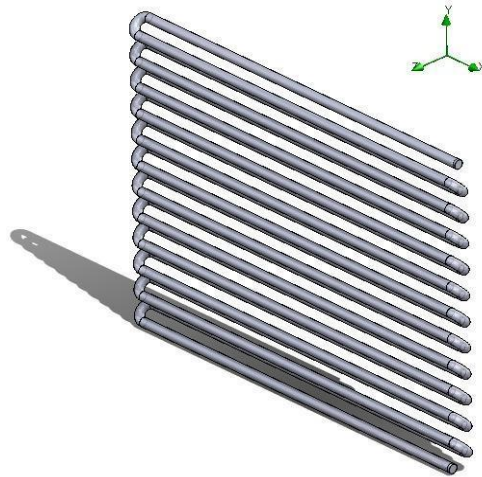
Due to its greater complexity, it is necessary to use local mesh refinement to ensure accurate results. Moving beyond the standard mesh model, intermediate and advanced refinement options available in Flow Simulation can be employed, as illustrated in Figures 6 and 7 below. These features allow for the enhancement of mesh quality in specific regions of the heat exchanger, ensuring a more precise and detailed analysis. Intermediate and advanced refinement are crucial steps to obtain reliable results.

Figure 6. Intermediate local refinement - three coordinates.
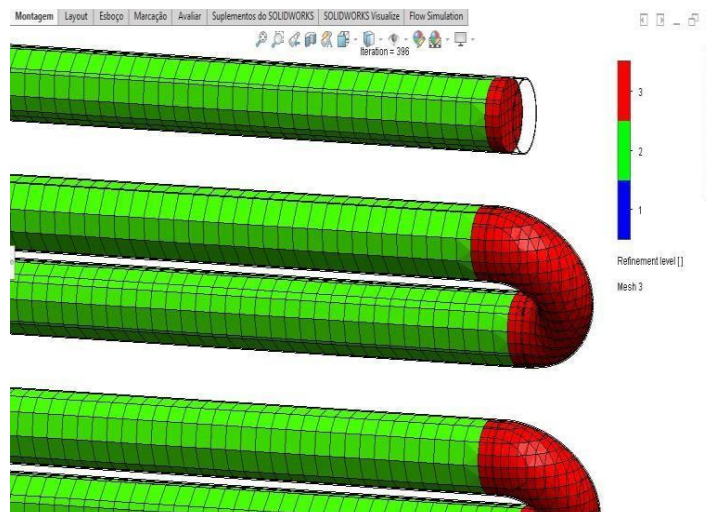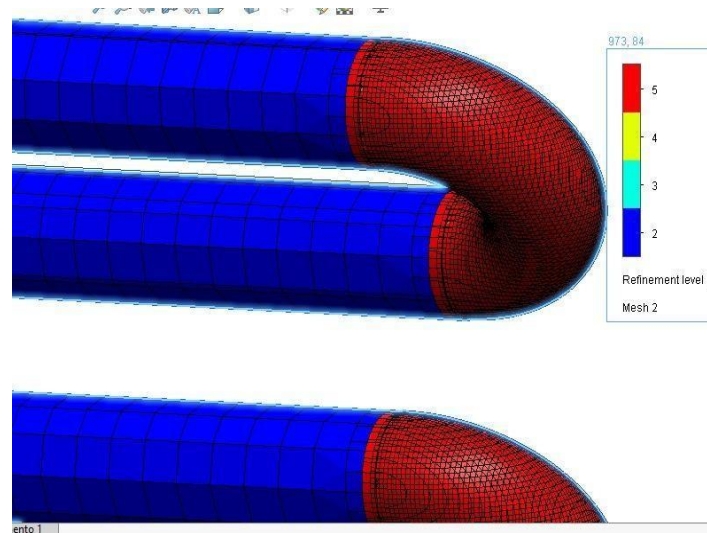

**Figure 7. High-level advanced mesh refinement - three coordinates.**

The results from the local refinements, conducted across the average gap of the analyzed temperatures, showed greater consistency with the analytical results when using the high level of refinement, compared to the intermediate refinement, as illustrated in Table 4.

Table 4. Comparison between results from different levels of local refinement.

Type of Local Refinement	Tube Temperature (°C)	Reynolds Number (Analytical)	Reynolds Number (Simulated)	Variation (%)
Intermediate	150	160406,41	172645,42	7,63
Advanced	150	160406,41	160898,14	0,55

With the optimal local refinement defined, it is possible to execute the same simulation process previously performed on a single tube, now applied to the entire coil, using the mesh with the best results at all temperatures for validation, as illustrated in Fig. 8 below:

Figure 8. Simulation of fluid heating in the heat exchanger coil.
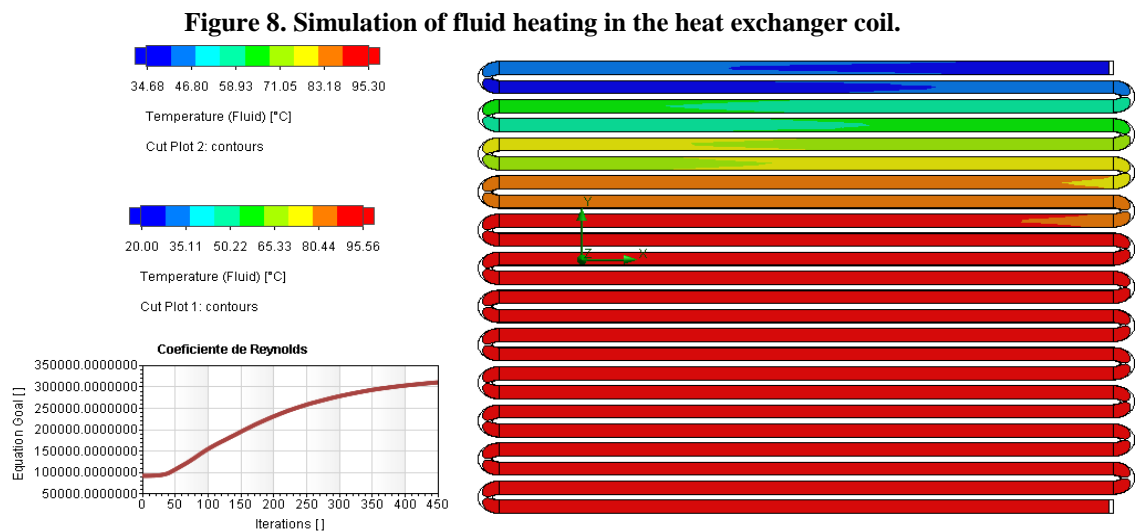
With the simulation performed on the entire coil, a significant increase in fluid heating can be observed due to the extended contact time with the heated tubes. Additionally, it is important to note that the Reynolds number is reduced due to pressure losses resulting from friction with the tubes and the bends in the coil. These results are illustrated in Table 5.

Table 5. Comparison of Results at Different Local Mesh Refinement Levels

Tubing Temperature (°C)	Tubing Temperature (°C)	Reynolds Number (Analytical)	Reynolds Number (Simulated)	Variation (%)
50	46,77	159601,90	159777,46	0,11
100	94,74	309713,49	310549,71	0,28
150	143,58	482669,06	485806,40	0,55
200	192,33	659786,74	664867,09	0,67
250	240,80	835958,36	842729,62	0,81

The results obtained revealed a deviation of 0.11% at a temperature of 50°C and 0.81% at a temperature of 250°C. These results provided a comprehensive validation for the entire proposed model. Additionally, it was observed that the number of tubes in the coil is effective, as the fluid heating achieved a maximum deviation when the tube reached 250°C and the fluid outlet recorded 240.80°C. The next and final step for validation is to compare the simulated results with the experimental data described in the article by Ma et al. (2020). This will allow for a more complete analysis and a more robust validation of the proposed model.

III.1.3 Experimental Validation

As illustrated in Table 2, the experimental process used by Ma et al. (2020) involves the refrigerant fluid being heated as it flows through the tubing and cooled by the air until its exit. The air is heated as it passes over the fins and through the entire assembly. The simulation process is performed with the same boundary conditions as those listed in Table 2, using the complete model of fins and mesh previously validated with the analytical method, as shown in Figure 9.

Figure 9. Fluid cooling simulation for experimental validation.
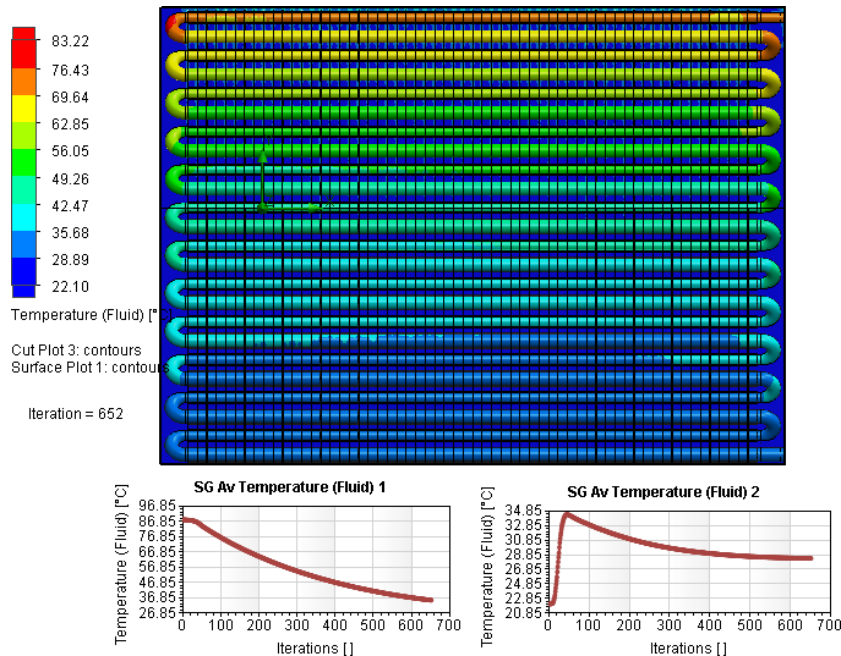
The results obtained from the simulation of each proposed case and the data from Table 2 are presented with their respective comparisons in Table 6.

Table 6. Comparison between results of different refinement levels.

Case	Hot Temperature (°C)	Variation (%)	Cold Temperature (°C)	Variation (%)
Case 1	53,38	11,8	35,24	13,4
Case 2	53,83	12,6	32,34	12,1
Case 3	53,47	9,75	32,23	12,5
Case 4	54,64	10,4	32,53	10,8

In this stage of the validation process, the comparison of results is focused on the outlet temperatures of the hot and cold fluids, rather than the Reynolds number as in the experimental validation. A maximum variation of 13.4% in the cold outlet temperature was observed in Case 1, and a minimum deviation of 9.75% in the hot outlet temperature was observed in Case 3. These results provided an acceptable deviation for comparison with the experimental results of the model.

IV. CONCLUSION

The main objective of this work was to create and validate a Computational Fluid Dynamics (CFD) model for a compact heat exchanger. This objective was successfully achieved. The work was divided into two stages. In the first stage, the proposed CFD model was constructed and validated using the Reynolds number as a validation criterion, with deviations of 0.11% at 50°C and 0.81% at 250°C. Subsequently, the model was also compared with experimental data collected from a previous study to ensure its consistency, with a maximum variation of 13.4% in the cold outlet temperature in Case 1 and a minimum deviation of 9.75% in the hot outlet temperature in Case 3. The model was successfully validated and proved capable of performing various simulations and optimizations for compact heat exchanger models.

The results obtained clearly demonstrate that applying this CFD model is beneficial in validating compact heat exchangers. It is highly interesting to consider expanding the use of the CFD model to validate other variables of the heat exchanger, as well as exploring its application in different types of heat exchangers. This approach would allow for the investigation of additional design aspects, such as fin placement and heat transfer surface design. With this, we could achieve substantial improvements in heat exchanger efficiency, contributing to enhanced performance and more efficient energy use. The expansion of the CFD model's use has significant potential for future studies and applications in thermal engineering.

Moreover, the main advantages of this method in thermal exchange equipment include reduced resources needed for development, decreased prototyping time, and a significant increase in productivity and effectiveness of these equipment. Therefore, in future work, it would be interesting to analyze this model with the aim of optimizing the thermal exchange of compact heat exchangers by varying fin designs, increasing the number of variables, and applications of the CFD model. Additionally, for further exploration, studies with other types of heat exchangers, such as shell-and-tube exchangers, could be conducted.

REFERENCES

[1]. BARBOSA, Nathalia Silvestrin et al. "Projeto de um protótipo de trocador de calor." *Revista Brasileira de Ciência, Tecnologia e Inovação*, vol. 2, no. 2, 2017, pp. 109-124.
[2]. Çengel, Yunus A.; Ghajar, A. J. Heat and Mass Tranfer, 4th Edition. 2012.
[3]. Incropera, Frank P. et al. Fundamentals of Heat and Mass Transfer.
[4]. Ma, X., Zhang, Q., Wang, J., & Yu, Y. (2020). A coupled CFD approach for performance prediction of fin-and-tube condenser. Numerical Heat Transfer, Part A: Applications, 78(6), 215-230.
[5]. Rodrigues, Anderson. Computational Fluid Dynamics (CFD): What Is It? What Is It For? Why Use It? Engineering and Architecture, 2019. Available at: http://www.engenhariaearquitetura.com.br/2019/09/dinamica-dos-fluidos-computacional-cfd-o-que-e-paraque-serve-porque-usar. Accessed: 23/04/2021.
[6]. Yousefi, Moslem et al. Multi-stage thermal-economical optimization of compact heat exchangers: a new evolutionary-based design approach for real-world problems. Applied Thermal Engineering, v. 83, p. 71-80, 2015.