SSRG International Journal of Mechanical Engineering
ISSN: 2348-8360/ https://doi.org/10.14445/23488360/IJME-V12I3P108

Volume 12 Issue 3, 93-108, March 2025

Original Article

Comparative Analysis of Numerical and CFD Approaches for Cooling Performance Evaluation in Tube and Plate Heat Exchangers

Girish Tukaram Panchal[1], Pravin T. Nitnaware[2], Ravikant K. Nanwatkar[3], Premendra J. Bansod[4]

[1,2]*Department of Mechanical Engineering, D.Y.Patil College of Engineering, Akurdi, Savitribai Phule Pune University, Pune, Maharashtra, India.*
[3]*Department of Mechanical Engineering, Sinhgad College of Engineering, Ambegaon, Savitribai Phule Pune University, Pune, Maharashtra, India.*
[4]*Department of Mechanical Engineering, G.H.Raisoni College of Engineering & Management, Wagholi, Savitribai Phule Pune University, Pune, Maharashtra, India.*

[1]*Corresponding Author : girishtpanchal@gmail.com*

Received: 13 January 2025 Revised: 12 February 2025 Accepted: 11 March 2025 Published: 29 March 2025

Abstract - *Tube and Plate category heat exchangers are widely used in industrial applications requiring efficient heat transfer. This research numerically and computationally analyzes their heat transfer characteristics by applying formulae from single-row, single-pass Tube and Fin heat exchangers. A numerical model evaluates the effects of varying inlet velocities of air and water, plate thickness and width, tube height and width, and inlet temperatures. A Python-based tool simulates six configurations simultaneously for quick predictions. CFD analysis is conducted using ANSYS Fluent Student Version 2024 R1 and R2 for validation. Key findings indicate that numerical predictions of outlet water and air temperatures are accurate to a minimum of 98.94% and 99.02%, respectively, compared to CFD. The highest outlet water temperature drop occurs at the least inlet water velocity (0.0625 m/s) and peak air velocity (24 m/s), with a temperature reduction of 10.5K in CFD and 11.56K in numerical calculations. Increased plate width enhances heat transfer, while reduced tube height results in lower outlet water temperatures. Inlet water temperature strongly affects the temperature drop, with the maximum difference observed at the lowest inlet air temperature. This study establishes a strong correlation between numerical and CFD results, enabling reliable preliminary design and optimization of Tube and Plate heat exchangers. The approach reduces computational time and resources while improving design efficiency.*

Keywords - *Tube and plate heat exchanger, Plain finned tube heat exchanger, Crossflow heat exchanger, CFD analysis, Heat transfer, e-NTU.*

1. Introduction

Heat exchangers are vital components in numerous industries, enabling heat transference amid fluids to maintain optimal operating temperatures and enhance system efficiency. They are crucial in applications like automotive, power production, chemical dispensation, Heat ventilation and air conditioning, and refrigeration systems. The primary types of heat exchangers include shell-and-tube, plate, finned tube, etc. Among them, Shell-and-tube exchangers are robust and suitable for high-pressure applications, while Tube and plate heat exchangers offer high efficiency with multiple thin plates. Tube and Plate exchangers increase the heat transfer surface area, enhancing performance.

A tube and plate heat exchanger consists of tubes with attached plates to increase surface area and improve heat transfer. These are commonly used in applications requiring efficient cooling, such as industrial radiators or heat exchangers. Their construction involves horizontal tubes and vertical plates, with fluids flowing perpendicularly, making them compact and efficient. Advantages include enhanced heat transfer and compact design, while demerits may involve higher manufacturing costs and potential for fouling.

Evaluating the cooling performance of crossflow heat exchangers involves numerical methods like the effective Number of Transfer Units (NTU) method and the Log Mean Temperature Difference (LMTD) approach. The effectiveness-NTU method focuses on the association between the heat exchanger's effectiveness and its heat transmission capacity, while the LMTD method uses temperature differences to determine heat transfer rates.

Computational Fluid Dynamics (CFD) software, such as ANSYS Fluent, COMSOL Multiphysics, and Open FOAM, provides detailed simulations of heat exchanger performance, offering insights into temperature distributions and fluid velocities. Full-scale CFD analysis requires significant resources, including licensing and powerful workstations. Developing a numerical calculator for initial calculations can help identify outlet water and air temperatures, providing a practical tool for preliminary evaluations before detailed CFD analysis.

CFD tools are accurate in forecasting the stream and heat transfer features, but large computing power is required, and the time required for setting up the multiple configurations and simulations is very high. For initial predictions or direction setting, it is necessary to make quick calculations with a good level of accuracy. To develop such, it is necessary to carry out numerical and CFD experimentation, and co-relation needs to be established. This research is needed to model the heat transfer characteristics in tube and plate-type heat exchangers and establish the co-relation with CFD simulations so that initial analysis of design direction can be decided based on tube and plate-type heat exchangers.

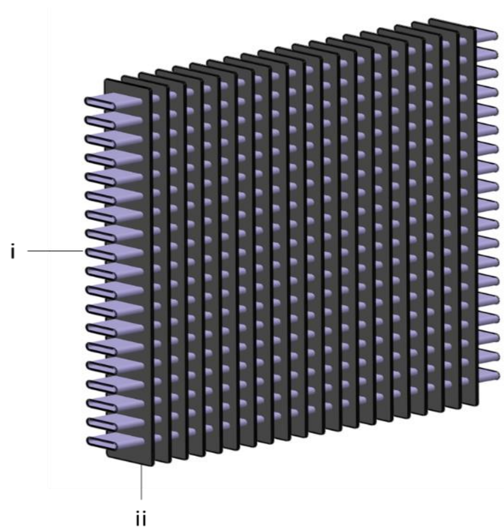


Fig. 1 General construction of tube and plate heat exchanger with (i) Tubes, and (ii) Plates.

2. Literature Review

This section focused on identifying the research gap and novelty of work by analyzing the literature considering plain-finned tube plate heat exchangers, numerical methods for heat transfer, E-NTU method for heat transfer, and CFD analysis of radiators.

Arjun Sastiya et al. worked on Heat exchanger simulation in Ansys Fluent for Enhancement of Radiator Efficiency Using CFD Analysis. According to the study, reducing tube diameter decreases mass flow rate and hence cooling capacity. However, this reduction also causes a reduction in the cooling rate.

R. Paul Linga et al. Researched "Design and Modification of Radiator in I.C. Engine Cooling System for Maximizing Efficiency and Life". It presents a way to increase engine cooling by adding a tapered nozzle, which lowers pressure, increases velocity, and complies with the ideal gas equation, improving radiator efficiency and overall performance.

P.S. Amrutkar et al. Worked on "Automotive Radiator Sizing and Rating Simulation Approach", showing that one-dimensional CFD software simulation findings and theoretical thermal analysis using the ε-NTU approach match closely. This suggests that 1-D simulation can be used to determine radiator core dimensions, considering thermal factors.

Badgujar Pankaj et al. Examined the "Analytical Performance Analysis of Cross Flow Louvered Fin Automobile Radiator." Experiments and GT model data were used to validate their suggested numerical strategy using the e-NTU method. They have developed an analytical model for a radiator with a rectangular tube and a louvered fin. The model predicts pressure drops, coolant and air outlet temperatures, and heat transfer rates with maximum variations of 2.75%, 3.29%, and 10.97%, respectively, and agrees with the results of both the GT and experimental models. Design optimization is made easier by this model's ability to accurately forecast radiator performance metrics.

K. Arasu et al. Verified the precision of the software HX combined in establishing radiator design parameters. Verifiable mistakes are found when comparing the 'HX combine' results with the manually computed results, suggesting that the program is highly accurate in radiator design. The study used the LMTD and ε-NTU methods to investigate the thermal behavior of car radiators, considering a number of geometrical characteristics and mass flow rates of coolant and air. The main way to improve radiator performance is to modify the convective heat transfer coefficient. Case examples show how effectively software-generated findings may be used to design radiators, saving designers valuable time. Furthermore, the comparison with case study findings shows that the ε – NTU method is the most dependable method for building crossflow type heat exchangers.

Paul O. Okhiria et al. Revealed a 2.4 percent discrepancy between the experimental results and the results of numerical calculations made using the ε – NTU approach. In addition, there is a 5.4% variation in the heat transmission rate. The theoretical technique is an accurate tool for radiator design because, despite small disparities, the experimental methodology eliminates mistakes due to ambient air oscillations.

Matthew Carl et al. A paper titled "An Investigative Analysis of Heat Transfer Processes in an Automobile Radiator" examined both theoretical and experimental aspects. Because internal water flow in the radiator is a closed system, their results show that the investigational heat transmission rate for water closely resembles the theoretically expected rate. On the other hand, the open-system dynamics of external airflow analysis lead to differences between theoretical and experimental results for outlet temperature and air heat transfer rates. The efficacy equation becomes precise only when the ratio of heat capacity to volume equals one. Nevertheless, the radiator's heat capacity ratio is 0.455. The discrepancy between theoretical and experimental values is caused by the efficiency equivalence, which is merely a good estimate in this particular case.

Ernest Palacios Surribas, in the article "Analysis of Automotive Thermal Management Heat Exchangers," described how simulation results vary as per air pressure drops, and heat transfer rates differ by 5% to 20% from actual data. Although these mistakes are considered acceptable, there are suggestions on reducing them, such as using different correlations, doing geometry calculations, or using different approaches than only ε-NTU. Calculations of liquid pressure drop show larger inaccuracies, which calls for a reexamination of the correlation. Consistent data in air pressure declines, outlet temperatures, and heat transfer validate the code's functionality. The thorough study of the thesis is strengthened by the effective simulation of various working situations, including heat transport directionality.

Ruchit Doshi et al. Mass flow rates of coolant and air velocities through a radiator are compared using various theoretical techniques in the research paper "Automobile Radiator Design and Validation". Particular focus was placed on engine RPMs and vehicle velocities used for the data collection, computation, simulation, and recording processes. The results show a close agreement between estimated and practical heat rejection at lower vehicle speeds, with a 5% difference from CFD results. However, when the vehicle speed rises, the expected heat rejection exceeds the real values by as much as 25%, possibly because of side pod flow separation.

Utkarsh Wani et al. Carried out a study entitled "CFD Analysis of Cross Flow Heat Exchanger and Experimental Validation" They performed a numerical analysis, verified by experimental data, using Ansys Fluent 18.1 on an elliptical tube blank in a crossflow heat exchanger. The results indicated agreement within error margins of 30 to 40% for a range of mass flow rates. The results of the temperature distribution investigation showed that the air outlet temperature varied by 2 to 3%. In certain Reynolds number ranges, the elliptical tube blank showed an enhancement ratio of 1.3 to 1.7 compared to circular blanks, delaying boundary layer separation and minimizing wake production. Furthermore, despite a lower pressure drop from eddy formation, orientation at a 45° angle promoted greater airflow mixing and higher heat transfer rates.

Khin Mar Koa et al. Worked on the impact of radiator height on heat exchanger effectiveness and heat transfer rate. In their research work "Design Analysis of Car Radiator in an Engine Cooling System," Raising the radiator's height improves the heat transmission rate and effectiveness. A thirty percent drop in height results in a nearly nineteen percent decrease in heat transfer rate. Fin distance decreases as the number of fins per column increases, yet the heat transmission rate increases. Adjusting the number of fins per column is necessary to maintain the same heat transfer rate with a lower radiator height. According to their investigation, the best fin arrangement guarantees the appropriate heat transfer rates at various radiator heights.

Manichander Rama et al., in the research paper "Design and Thermal Analysis of an Automotive Radiator for Enhancing Flow Uniformity using CFD", worked on comparing a three-pass radiator to a single-pass one; they discovered appreciable gains in thermal performance and flow uniformity. The three-pass design improved flow homogeneity, extended radiator life, and decreased corrosion by lowering flow rate variance by 57%. The three-pass radiator's thermal analysis showed a 42% increase in heat transfer rate and a 9°C lower temperature drop. Fin geometries will be investigated in future studies to further maximize radiator efficiency.

José Canazas et al. the paper entitled "Heat Transfer and Pressure Drop Performance of a Hydraulic Mining Shovel Radiator Using Ethylene Glycol/Water-Based Al2O3 Nanofluids" carried out numerical analysis In order to assess radiator performance in heavy-duty circumstances. The results showed shortcomings in the current radiator design and recommended changes to the coolant-side and air-side geometries. They suggested using copper for radiators to improve maintenance viability. The study focused on analysing engine and cooling system dynamics in conjunction with nanofluids heat transfer, which is important for equipment such as hydraulic mining shovels. They also mentioned the possibility of decreasing pumping power and pressure decrease in laminar and transitional circumstances. Future research should think about creating specialized test benches that are suited to the operational and environmental circumstances unique to these heat exchangers.

M.K. Chopra et al. researched the topic of "Performance Analysis of Heat Exchanger Using Different Materials by CFD at Different Mass Flow Rate of Water and Air Velocity" Their research entails creating a radiator-type heat exchanger Computational Fluid Dynamics (CFD) model using experimental data. They examine the effects of different parameters on heat transfer rates using numerical analysis. Higher heat transmission is found in finned tube configurations when comparing radiator tubes with and without fins. Furthermore, they find that when hot water's mass flow rate increases, the heat transfer rate decreases, whereas higher air velocities promote heat transfer and expedite water cooling. Additionally, their investigation finds that fin and copper tube radiators are better at transferring heat than brass and aluminum radiators.

Jeevananth P et al. worked on heat transfer analysis on an automotive radiator utilizing ethylene glycol as a coolant and air speeds ranging from 15 kmph to 75 kmph. The results indicate that when the air's Reynolds number grows, the Nusselt number rises by 69% to 125%. The heat transfer coefficient rises by 125% at increasing air velocities. However, as the Reynolds number increases from 14,000 to 71,000, fin efficiency reduces by 6.1%. In the same circumstances, there is a 91% increase in the overall heat transfer coefficient. The radiator functions well at higher speeds, providing sufficient cooling even when engine heat rises, despite a minor decrease in fin efficiency.

Wang C. et al. worked on A correlation for fin and tube heat exchangers with plain fin geometry in the article titled, 'Heat Transfer and Friction Characteristics of Plain Fin-and-Tube Heat Exchangers, Part II: Correlation' the paper presented correlation for heat transfer also includes the contact conductance and 88.6% of the data are within ±15%; the friction correlation corresponds to 85.1% of the database within the same range. The average absolute deviations of the heat transfer and friction can be determined as 7, 53% and 8, 31 %, respectively. The investigation made in this study concerns the geometric parameters by mathematical modeling.

Basavarajappa S. et al., in the research article "A Review on Performance Study of Finned Tube Heat Exchanger", discussed various fin geometries for the purpose of heat transfer enhancement. Rectangular, triangular, trapezoidal, pin, wavy, offset strip, louvered and perforated fin types were compared to determine the fin's heat transfer rate and pressure drop. These were the fin pitch, orientation, height, and groove patterns, and the results were analyzed using the Nusselt number, the friction factor, and Rayleigh numbers. The findings also show that fins enhance heat transfer since they create a large surface area to area to expose to the hot air as well as enhance the mixing of the fluid. Nevertheless, while the standard wavy and rectangular fins gave better heat transfer characteristics, they were associated with a higher delta pressure.

Saichandar G. et al. Discussed the heat transfer and friction aspects of different types of fins in the paper titled "Design and Analysis on Compact Fin Heat Exchangers with Perforations". This paper carried out CFD analysis on the plain fin, strip offset fin, circular perforated fin, and elliptical perforated fin based on parameters of fin thickness, spacing, and within thermal entry length at Re 285. Recent studies reveal that using perforated fins improves both conductance and frictional factors by adopting plain and perforated fins, where the elliptical shape surpasses the circular shape. Plain fins and offset fins showed more heat transfer and friction than conventional fins, and out of these two, offset fins with elliptical perforations had the highest power because they made a larger surface area and turbulence. The contents also explain the consequences of perforations and offsets on heat exchangers' performance despite the pressure drop increase.

Zhang L. et al., in the "Numerical Study of Fin-and-Tube Heat Exchanger in Low-Pressure Environment" study, presented the performance of a plain fin-and-tube heat exchanger under low-pressure environments. The work employed CFD analysis to investigate heat change, pressure drop and entropy at various air pressures. The recorded results indicated that both HTC and pressure drop decreased by 67.92% and 53.45% for the value of 25 kPa compared to 101 kPa. Entropy generation increased from 0.006 to 0.018, that is, by 205.8%, when the air pressure increased from 25 kPa to 101 kPa due to greater temperature differences than pressure drop. Analysis was carried out on the preliminary data from the rough wall using trendlines to predict the heat transfer and friction factors, and it yielded high accuracy with a mean absolute error of 7.48% for the heat transfer factor and 9.42% for the friction factor.

Abbas A.S. et.al. The research paper "Enhancement of Plate-Fin Heat Exchanger Performance with the aid of (RWP) Vortex Generator" discussed the heat transfer enhancement of PFHEs with OSFs. OSFs help increase the surface area for heat transfer, eliminate the thermal boundary layer by disrupting its formation, as well as helping induce secondary flows. It is established that the various types of OSFs provide a higher heat transfer than plain fins at Re, which is less than 1000, while plain fins are superior at high Re. With reference to j and f, the study deduced these two factors, where Re increase decreases fin pitch, thus the heat transfer to friction ratio.

Jafari Nasr et al. Worked on Low-finned tube heat exchangers. Where the work endeavored to put forth the thermal and hydraulic performance correlations of low-finned tube heat exchangers, it was ascertained that despite the higher dispersion of ESDU correlations, the associated accuracy is still the highest. The study supported research done by the authors where the efficiency of heat ex-tension is improved by using low-finned tubes instead of plain tubes, coupled with the fact that the number of shells needed in a particular duty is also reduced.

Wang, Y.-Q. et al. undertook a numerical analysis of plate-fin heat exchangers with plain and serrated fins under low Reynolds numbers. A relatively easy-to-implement numerical model was created, and proof of the model's accuracy was obtained through comparison with experimental values, with a good agreement in demonstrating heat transfer as well as the flow characteristics of the fluid. Thus, it increased confidence in the possibility of using this model for further analysis and optimization of similar heat exchanger structures.

Stewart, S. et al. Conducted a study to compare plainly finned and louvered finned-tube condenser heat exchangers of residential air conditioning systems. They utilized a thermo system model and optimization technique; they noted that the optimized louvered fin enhanced the heat exchanger efficiency by 6.15% and decreased the cost of the material by 56% for the same COP. He also identified that if louvers are added without re-optimization, then it can reduce efficiency by 2.2 percent to 6.1 percent.

Dhangar, I.J. et al. Have selected five fin-and-tube heat exchangers, and the experimental studies have been carried out on the air side at the Re number ranging from 4000 to 10000. Prior to the optimization, the best heat transfer was observed on slit fins at high Reynolds numbers.

Heat exchangers with the general configuration of the vortex generators used in the study worked best at larger attack angles, overall length, and smaller heights. When the Reynolds number was increased, pressure drop was high, and outlet temperature was low, which showed that flow disturbance and heat transfer were more affected.

Basavarajappa et al. discussed a wide variety of fin types, such as rectangular, triangular, trapezoidal, pin, wavy fin, offset strip fin, louvered fin, and perforated fins, in terms of heat transfer rate and pressure drop. Some parameters that were taken into account were fin pitch, orientation, height, and groove patterns.

The study concluded that fins are beneficial in this case as they increase surface area and promote turbulence. Wavy and rectangular fins are more effective in terms of heat transfer rates, but at the same time, they yield higher pressure drop rates.

The available research literature indicates that there is still a significant opportunity for developing a correlation between Numerical calculation and CFD analysis of cooling performance of Tube and Plate type crossflow Heat exchanger. By further research into this area, accurate mathematical models and methods of CFD simulation can be developed for improving the performance of crossflow heat exchangers.

3. Methods

3.1. Simplified representation of Single Row Tube & Plate Heat Exchanger and Parameters Related to It

A simple example of a Tube and Plate Heat exchanger is created and shown in Figure 2, which has 5 horizontal tubes and 5 vertical plates. Water flows through the tubes, and air flows across the tubes.

Table 1. Components in tube and plate heat exchanger

Sr.No.	Name of Component	Material	Quantity
i	Tube	Aluminum	5
ii	Plate	Aluminum	5
iii	Water	Water	As per input parameters
iv	Air	Air	

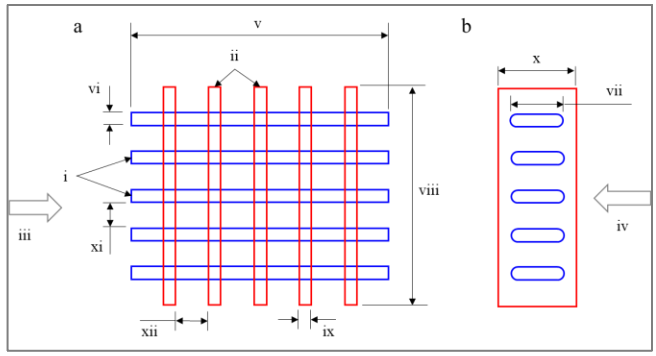


Fig. 2 Simplified representation of single row tube & plate heat exchanger (a) Front view, and (b) Side view.

Table 2. Parameters in tube and plate heat exchanger

Sr.No.	Description of Parameter
v	Length of Tube
vi	Height of Tube Section
vii	Width of Tube Section
viii	Length of Plate
ix	Thickness of Plate
x	Width of Plate
xi	Gap between Tubes
xii	Gap between Plates

Table 3. Input parameters related to hot fluid (water)

Sr. No.	Description of Parameter	Term used	Value	Unit
1	Velocity of Water at Inlet	Vw	0.0063	m/s
2	Temperature of Water at Inlet	Twi	355.15	K
3	Specific Heat of Water at Inlet	Cpw	4182	J/kg.K
4	Density of Water	Rho_w	998.2	kg/m³
5	Thermal Conductivity of Water	kw	0.6	W/m.K
6	Viscosity of Water	mu_w	0.001003	Pa.s
7	Prandtl number of Water	Prw	mu_w * Cpw / kw	

Table 4. Input parameters related to cold fluid (air)

Sr. No.	Description of Parameter	Term used	Value	Unit
1	Velocity of Air at Inlet	Vw	12	m/s
2	Temperature of Air at Inlet	Tai	297.15	K

3	Specific Heat of Air	Cpa	1006.43	J/kg.K
4	Density of Air	Rho_a	1.225	kg/m³
5	Thermal Conductivity of Air	ka	0.0242	W/m.K
6	Viscosity of Air	mu_a	1.79E-05	Pa.S
7	Prandtl number of Air	Pra	mu_a * Cpa / ka	

Table 5. Input parameters related to tubes

Sr. No.	Description of Parameter	Term used	Value	Unit
1	Length of Tube	Tl	0.25	m
2	Height of Tube section	Th	0.02	m
3	Width of the Tube section	Tw	0.05	m
4	Thickness of Tube	Tt	0.0005	m
5	Thermal Conductivity of Tube	kt	202.4	W/m.K
6	Number of Tubes	Nt	5	nos.

Table 6. Input parameters related to plates

Sr. No.	Description of Parameter	Term used	Value	Unit
1	Thickness of Plate	Pt	0.005	m
2	Height of Plate	Ph	0.23	m
3	Width of Plate in Side view	Pw	0.055	m
4	Thermal Conductivity of Plate	kp	202.4	W/m.K
5	Number of Plates	Np	5	nos.

3.2. Numerical Methods: e-NTU Formulae

3.2.1. E-NTU Formulae

The following equations are used to numerically calculate cooling performance.

$$Opt = (\pi * Th) + (Tw - Th) \tag{1}$$

$$Wpt = \left(\pi * (Th - 2 * Tt)\right) + \left(2 * \left((Tw - 2 * Tt) - (Th - 2 * Tt)\right)\right) \tag{2}$$

$$At = Nt * Opt * Tl \tag{3}$$

$$Ati = Nt * Wpt * Tl \tag{4}$$

$$Act = (Tw - (2 * Tt) - Th) * \left(Th - (2 * Tt)\right) + \left(\pi * \left(\frac{Th - Tt}{2}\right)^2\right) \tag{5}$$

$$Acto = (Tw - Th) * (Th) + \left(\pi * \left(\frac{Th - Tt}{2}\right)^2\right) \tag{6}$$

$$Ato = At - (Np * Nt * 2 * Acto) \tag{7}$$

$$Ap = Np * \left(2 * \left((Pw * Ph) + (Pt * Ph) + (Pw * Pt)\right)\right) - (Np * Nt * 2 * Acto) \tag{8}$$

$$Ar = At + Ap \tag{9}$$

$$St = \left(\frac{Ph}{Nt + 1}\right) - (Th) \tag{10}$$

$$Sp = \left(\frac{Tl}{Np + 1}\right) - (Pt) \tag{11}$$

$$Apa = \left((St * Sp) * \left(\left((Nt - 1) * (Np - 1)\right) + \left(2 * (Np - 1) * \frac{3}{4}\right) + \left(2 * (Nt - 1) * \frac{3}{4}\right) + 2\right)\right) \tag{12}$$

$$Awpa = \left(2 * (St + Sp)\right) * \left(\left((Nt - 1) * (Np - 1)\right) + \left(2 * (Np - 1) * \frac{3}{4}\right) + \left(2 * (Nt - 1) * \frac{3}{4}\right) + 2\right) \tag{13}$$

$$Dht = \frac{(4 * Act)}{Wpt} \tag{14}$$

$$Dha = \frac{(4 * Apa)}{Awpa} \tag{15}$$

$$Rew = \frac{(rho_w * Vw * Dht)}{mu_w} \tag{16}$$

$$Rea = \frac{(rho_a * v_a * Dha)}{mu_a} \tag{17}$$

$$if\ Rew < 2300\ and\ 0.5 < Prw < 100$$
$$Nuw = 4.36 \tag{18}$$

$$if\ 2300 < Rew < 10^4 and\ 0.5 < Prw < 2000\ and\ \left(\frac{Tl}{Dht}\right) > 10$$
$$f = (0.79 * \log(Rew) - 1.64)^{-2} \tag{19}$$

$$Nuw = \frac{\left(\left(\frac{f}{8}\right) * (Rew - 1000) * Prw\right)}{\left(1 + 12.7 * \left(\frac{f}{8}\right)^{0.5} * \left(Prw^{\frac{2}{3}} - 1\right)\right)} \tag{20}$$

$$if\ 10^4 < Rew < 5 * 10^6 and\ 0.5 < Prw < 2000\ and\ \left(\frac{Tl}{Dht}\right) > 10$$

$$f = (0.79 * \log(Rew) - 1.64)^{-2} \tag{21}$$

$$Nuw = \frac{\left(\left(\frac{f}{8}\right) * Rew * Prw\right)}{\left(1.07 + 12.7 * \left(\frac{f}{8}\right)^{0.5} * \left(Prw^{\frac{2}{3}} - 1\right)\right)} \quad (22)$$

$$if\ Rew > 10^4 and\ 0.7 < Prw < 160\ and \left(\frac{Tl}{Dht}\right) > 60$$

$$Nuw = 0.023 * Rew^{0.8} * Prw^{\frac{1}{3}} \quad (23)$$

$$if\ Rea < 5e^5$$

$$Nua = 0.664 * (Rea^{0.5}) * \left(Pra^{\frac{1}{3}}\right) \quad (24)$$

$$if\ 5e^5 \leq Rea < 1e^7$$

$$Nua = 0.037 * (Rea^{0.8}) * \left(Pra^{\frac{1}{3}}\right) - 871 \quad (25)$$

$$if\ Rea \geq 1e7$$

$$Nua = 0.036 * (Rea^{0.8}) * \left(Pra^{\frac{1}{3}}\right) \quad (26)$$

$$hw = Nuw * \frac{kw}{Dht} \quad (27)$$

$$ha = Nua * \frac{ka}{Dha} \quad (28)$$

$$m = \sqrt{\left(\frac{2 * ha}{kp * t}\right)} \quad (29)$$

$$Lc = \frac{St}{2} + \frac{Pt}{2} \quad (30)$$

$$nf = \frac{\tanh(m * Lc)}{m * Lc} \quad (31)$$

$$n0 = 1 - \left(\left(\frac{Ap}{Ar}\right) * (1 - nf)\right) \quad (32)$$

$$UA = (1 / (\left(\frac{1}{(n0 * ha * Ar)}\right) + \left(\frac{Tt}{kt}\right) + \left(\frac{1}{(hw * Ati)}\right) \quad (33)$$

$$Cw = rho_w * Vw * Act * Nt * Cpw \quad (34)$$

$$Ca = rho_a * Va * Tl * Ph * CPa \quad (35)$$

$$Cmin = \min(Cw, Ca) \quad (36)$$

$$Cmax = \max(Cw, Ca) \quad (37)$$

$$Cr = \frac{Cmin}{Cmax} \quad (38)$$

$$NTU = \frac{(UA)}{Cmin} \quad (39)$$

$$e = 1 - e^{\left(\left(\frac{1}{Cr}\right) * (NTU^{0.22}) \left(e^{\left(-Cr * (NTU^{0.78})\right)} - 1\right)\right)} \quad (40)$$

$$Q = e * Cmin * (Twi - Tai) \quad (41)$$

$$Two = Twi - \left(\frac{Q}{Cw}\right) \quad (42)$$

$$Tao = Tai + \left(\frac{Q}{Ca}\right) \quad (43)$$

Table 7. Parameter description

Sr. No.	Variable Parameter	Constant Parameter
1	Outer perimeter of one tube	Opt
2	Wetted perimeter of one tube	Wpt
3	Outer Surface area of all Tubes	At
4	Inner Surface area of all Tubes	Ati
5	Cross-sectional area of one tube inside	Act
6	Cross-sectional area of one Tube outside	Acto
7	Outer Surface area of all Tubes without plates	Ato
8	Surface area of all plates	Ap
9	Total surface area of the radiator	Ar
10	Spacing between tubes	St
11	Spacing between plates	Sp
12	Passage Area of Air	Apa
13	Wetted perimeter of the air	Awpa
14	Hydraulic Diameter of Tube	Dht
15	Hydraulic Diameter of Air	Dha
16	Reynolds Number for Water	Rew
17	Reynolds Number for Air	Rea
18	Nusselt number for Water flowing inside tubes if Re_water < 2300 and 0.5 < Pr_water < 100	Nuw
19	Nusselt number for air flowing across Tubes if Re_air < 3.5x10^5	Nua
20	Heat transfer coefficient of Water	hw
21	Heat transfer coefficient of air	ha
22	Coefficient for calculating efficiency	m
23	Corrected fin length	Lc
24	Fin efficiency	nf
25	Overall surface efficiency	n0
26	Overall heat transfer coefficient	UA
27	Heat capacity rate of coolant	Cw
28	Heat capacity rate of air	Ca
29	Minimum Heat capacity rate	Cmin
30	Maximum Heat capacity rate	Cmax
31	Heat Capacity Ratio	Cr
32	Number of Transfer Units	NTU
33	Effectiveness (ε) for crossflow heat exchanger with both fluids unmixed	ε
34	Heat transfer rate	Q
35	Temperature of Coolant at the outlet	Two
36	Temperature of Air at the outlet	Tao

3.3. Software Coding: Python is Used to do Simultaneous Calculations of Various Cases

- Python code is written and simulated in Pycharm and Jupyter Notebook IDE
- Six input values of water inlet velocity can be simulated at a time
- GitHub Repository https://github.com/Garyp36/Python-Code-for-Simple-Tube-Plate-type-Crossflow-Heat-Exchanger/blob/main/Code_250225

3.4. CFD Methods: Ansys Fluent

3.4.1. Steps Used for Preparing Geometry (Figure 3)

- Sketches for the first Tube and Air are prepared on XY plane.
- The first plate is prepared on the offset plane and then moved to the YZ plane.
- The remaining 4 Tubes and 4 plates are prepared using the pattern command.
- Tube bodies are subtracted from Plate bodies.
- Tube and Plate bodies are subtracted from the Air Domain.
- Solid property is applied for Plate bodies.
- Fluid property is applied for Air and Tube Bodies.
- Tube, Air and Plate bodies are selected together, forming a part with 11 bodies.
- Share topology is created for the formed part so that the Mesh will be interconnected.

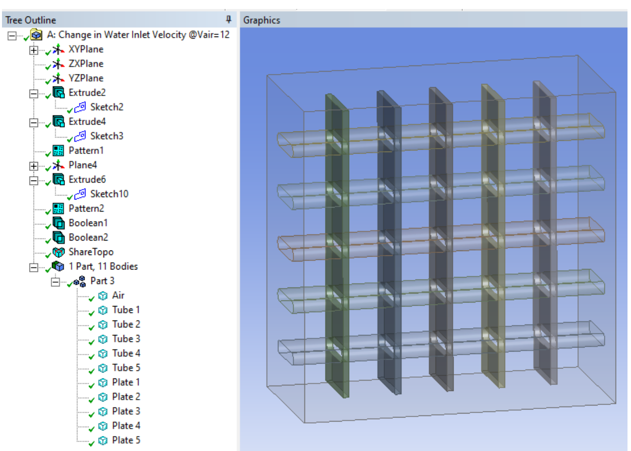


Fig. 3 3D modelling in design modeler of ansys fluent 2024R2

3.4.2. Steps Used for Meshing the Geometry

- Body Sizing of 0.0025m is applied to Tube and Plate Bodies
- Body Sizing of 0.006m is applied to Air Body
- Edge Sizing of 0.01m is applied to all edges of the Air Domain
- Inflation with a Smooth Transition of 3 layers with a growth rate of 1.2 is applied to Tube Walls
- Face Meshing is applied to walls perpendicular to the flow of air

- Named selections are created for Air Inlet, Air Outlet, Water Inlet, Water Outlet, Pipe Walls, Plates, Water Domain and Air Domain
- Meshing Quality is checked using Mesh Quality Worksheet Figure 5 and Mesh Metrics Figure 6.
- Statistics show the number of nodes and elements in Figure 7.

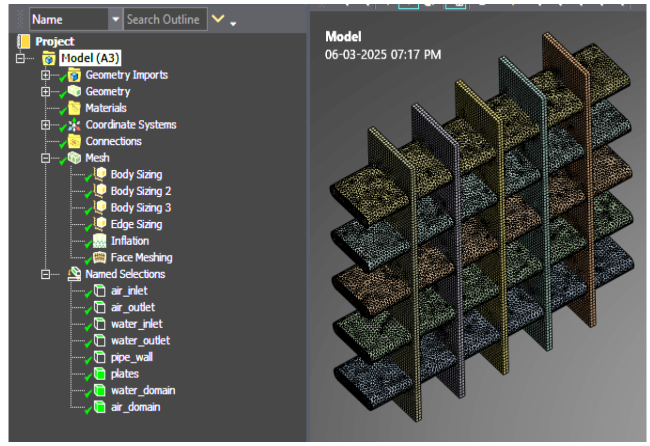
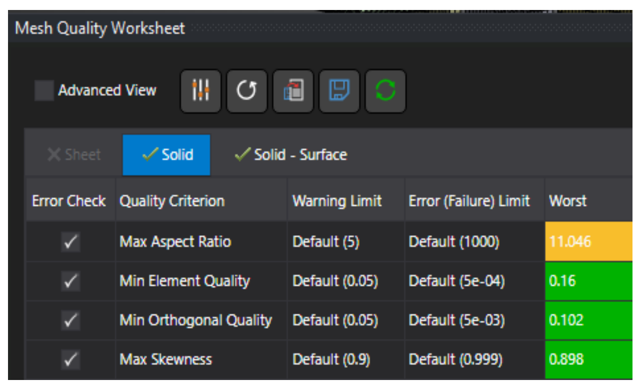


Fig. 4 Meshing in ansys fluent 2024R2



Error Check	Quality Criterion	Warning Limit	Error (Failure) Limit	Worst
✓	Max Aspect Ratio	Default (5)	Default (1000)	11.046
✓	Min Element Quality	Default (0.05)	Default (5e-04)	0.16
✓	Min Orthogonal Quality	Default (0.05)	Default (5e-03)	0.102
✓	Max Skewness	Default (0.9)	Default (0.999)	0.898

Fig. 5 Mesh quality worksheet

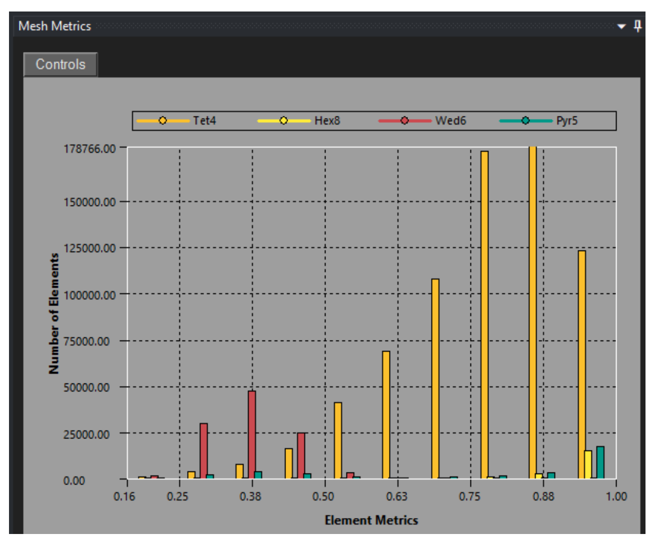


Fig. 6 Mesh metrics

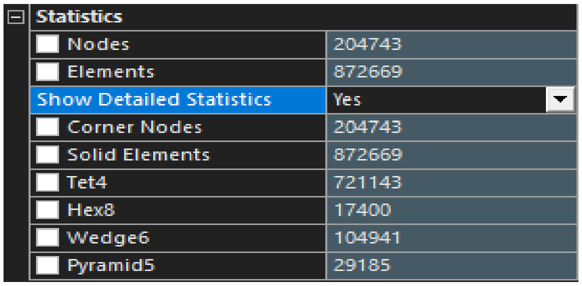


Fig. 7 Mesh statistics

Table 8. Setup parameters

Parameters	Sub Parameters	Value
Setup	Solver Option	Double Precision
	Solver Processes	4
General	Solver Type	Pressure-Based
	Solver Time	Steady
	Gravity	0
Models - Viscous Model	Energy	On
	Model	Laminar
Materials	water-liquid	
	air	
	aluminum	
Cell Zone Conditions	air_domain	air
	plates	aluminum
	water_domain	water-liquid
Boundary Conditions	air_inlet	velocity inlet 12m/s, 297.15K
	air_outlet	pressure outlet
	water_inlet	velocity inlet 0.0625m/s, 355.15K
	water_outlet	pressure outlet
	Pipe Wall	Coupled 0.0005m thk
Methods	Scheme	Coupled
	Gradient	Least Square Cell-Based
	Momentum	Second Order Upwind
	Energy	
Report Definitions	Outlet Air Temp Outlet WaterTemp	Area Weighted Average Temperature
	Heat Transfer	Total Heat transfer rate
Monitors	Residual	1.00E-06
	Convergence	Total Heat Transfer 0.1
Initialization	Method	Hybrid
Run Calculation	Number of Iterations	500
	Reporting Interval	1
	Profile Update Interval	1

Total Heat Transfer is ensured to be below 10% of Total Heat Transfer after the simulation is complete.

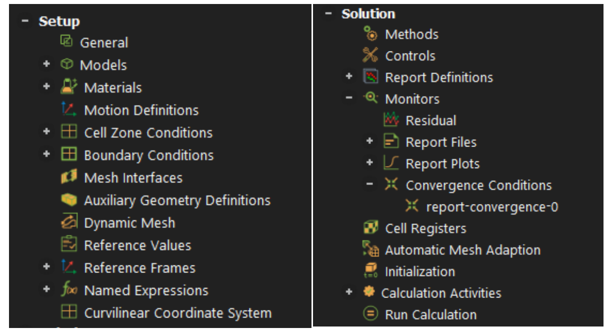


Fig. 8 Setup and solution parameters from ansys fluent 2024R2

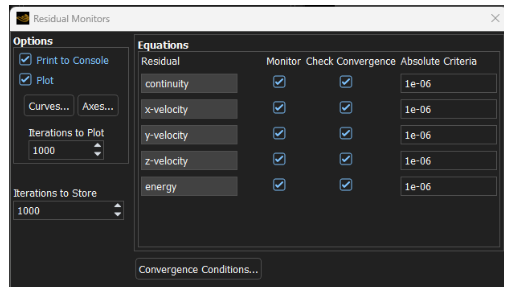


Fig. 9 Residual monitors

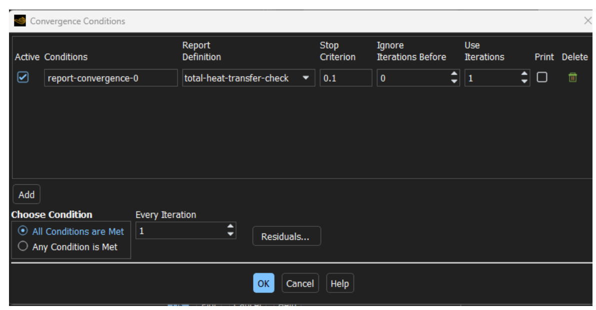


Fig. 10 Convergence conditions

3.4.3. Steps Used for Reading the Results (Figure 11)

- A function calculator is used to get the equivalent expression.
- An equivalent expression is tabulated in the table, and results are displayed.

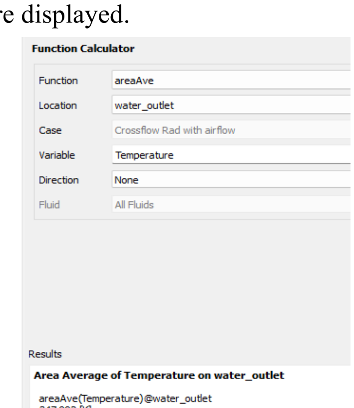


Fig. 11 Reading the results

3.4.4. Input and Output Parameters

- Variables of the design of experiments are identified as input parameters, and a design point table is used to run the simulations.
- Output parameters in each design point are monitored.

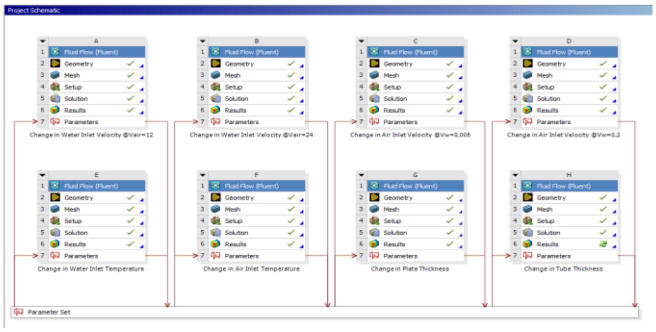


Fig. 12 Project schematic in ansys fluent 2024R2

To verify the effectiveness of the Numerical and CFD methods, following the design of experiments planned.

Table 9. Design of experiments

Sr. No.	Variable Parameter	Constant Parameter
1	Inlet Water Velocity	Inlet Air Velocity (lower and upper side)
2	Inlet Water Velocity	
3	Inlet Air Velocity	Inlet Water Velocity (lower and upper side)
4	Inlet Air Velocity	
5	Thickness of Plate	Inlet Water & Air Velocity
6	Width of Plate	
7	Height of Tube	
8	Width of Tube	
9	Water Temperature	
10	Air Temperature	

4. Results and Discussion

4.1. Results

The following are the graphs showing the results of the design of experiments. Outlet Water and Air temperature are monitored in each experiment.

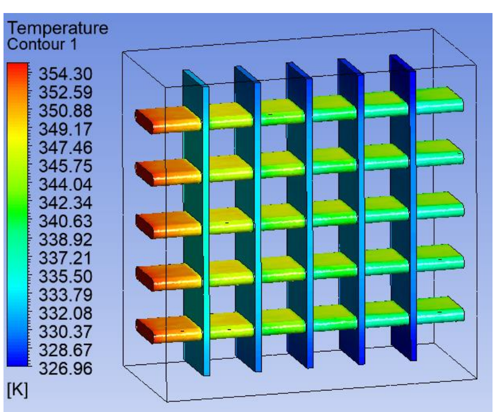


Fig. 13 Simulation results

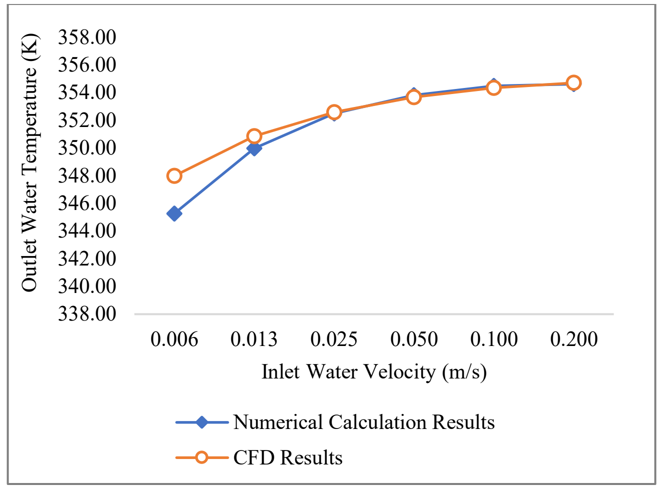


Fig. 14 Effect of change in 'inlet water velocity' and constant 'inlet air velocity' at 12m/s on 'outlet water temperature'

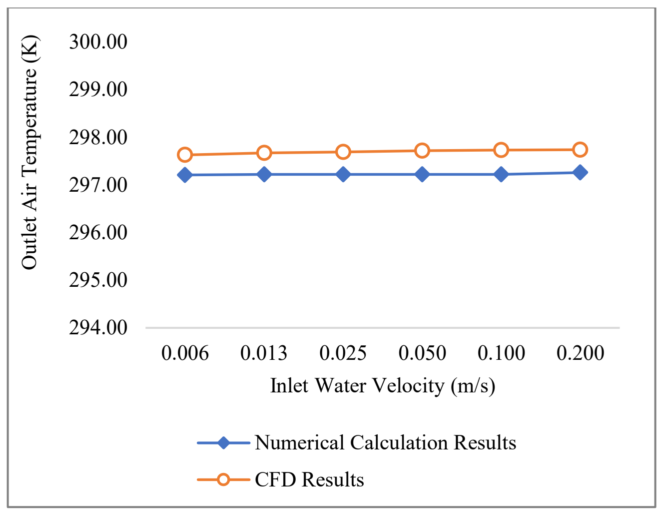


Fig. 15 Effect of change in 'inlet water velocity' and constant 'inlet air velocity' at 12m/s on 'outlet air temperature'

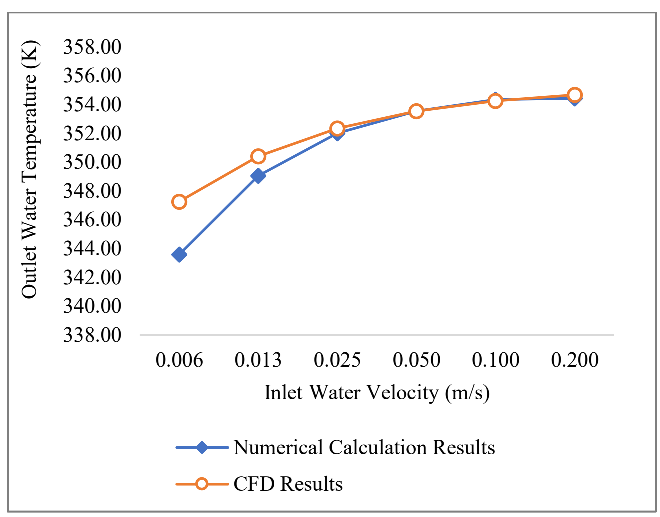


Fig. 16 Effect of change in 'inlet water velocity' and constant 'inlet air velocity' at 24m/s on 'outlet water temperature'

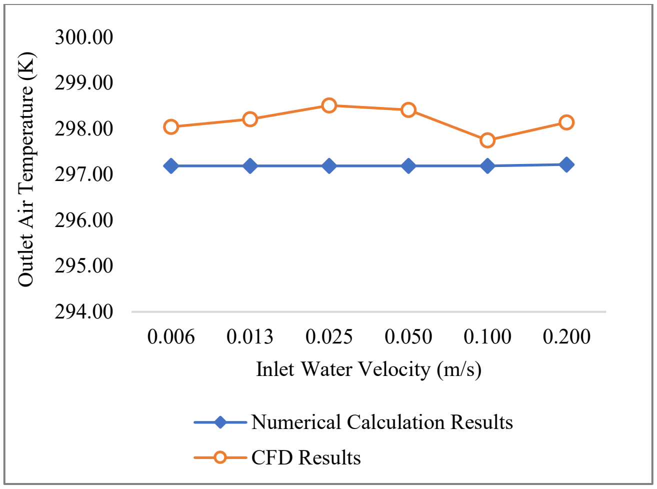


Fig. 17 Effect of change in 'inlet water velocity' and constant 'inlet air velocity' at 24m/s on 'outlet air temperature'

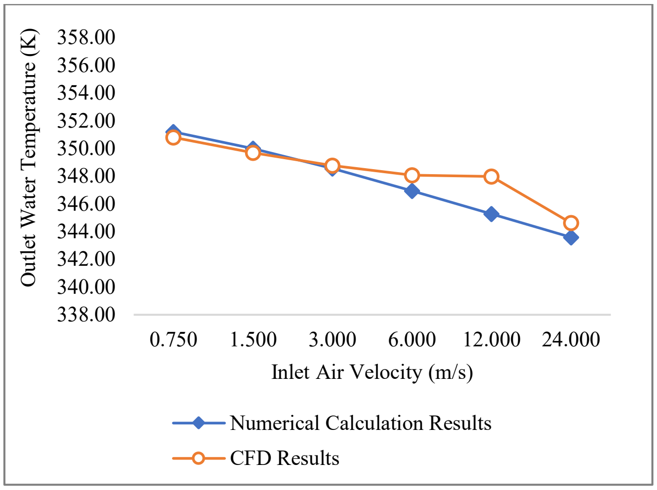


Fig. 18 Effect of change in 'inlet air velocity' and constant 'inlet water velocity' at 0.00625m/s on 'outlet water temperature'

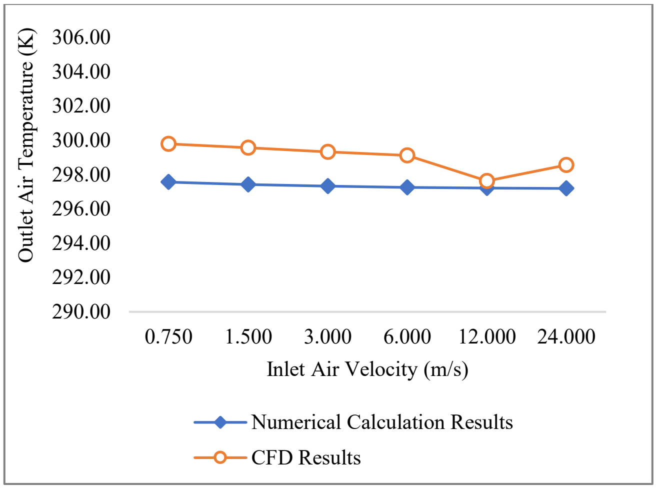


Fig. 19 Effect of change in 'inlet air velocity' and constant 'inlet water velocity' at 0.00625m/s on 'outlet air temperature'

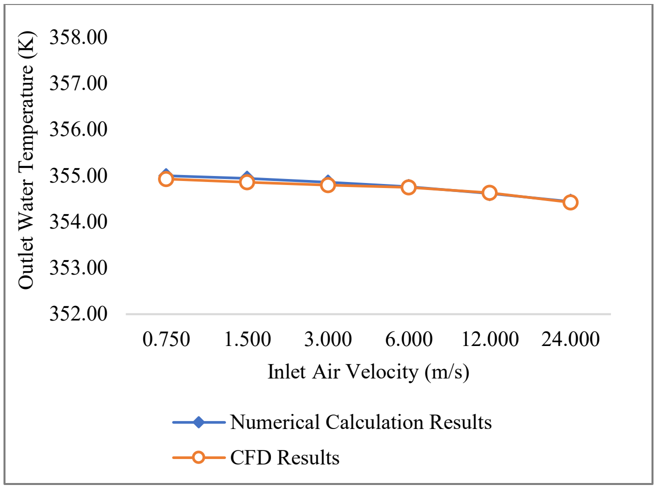


Fig. 20 Effect of change in 'inlet air velocity' and constant 'inlet water velocity' at 0.2m/s on 'outlet water temperature'

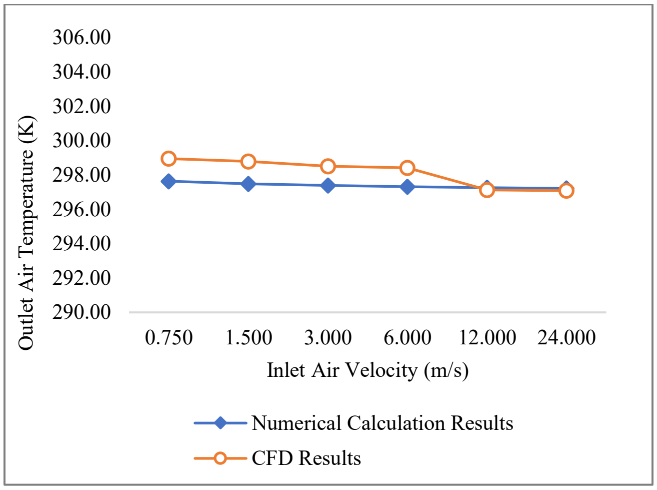


Fig. 21 Effect of change in 'inlet air velocity' and constant 'inlet water velocity' at 0.2m/s on 'outlet air temperature'

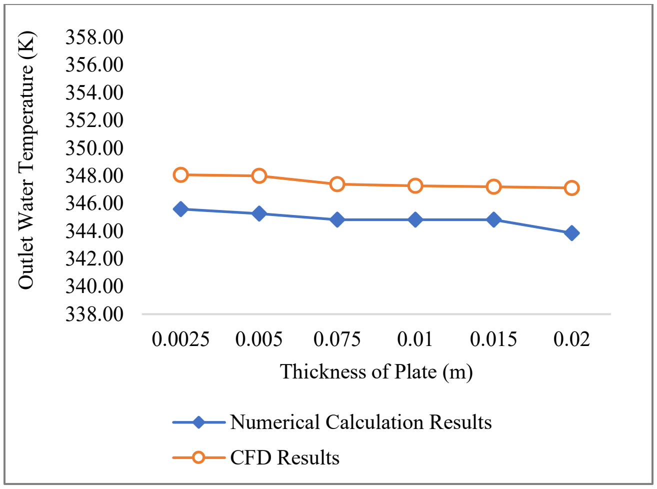


Fig. 22 Effect of change in 'thickness of plate' at constant 'inlet water velocity' of 0.00625m/s and constant 'inlet air velocity' of 12m/s on 'outlet water temperature'

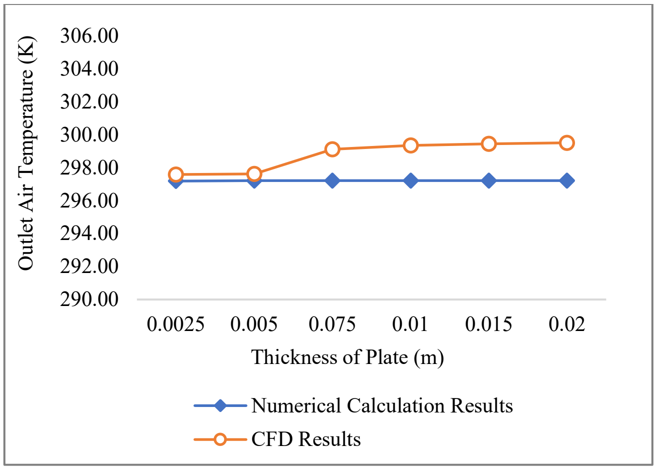


Fig. 23 Effect of change in 'thickness of plate' at constant 'inlet water velocity' of 0.00625m/s and constant 'inlet air velocity' of 12m/s on 'outlet air temperature'

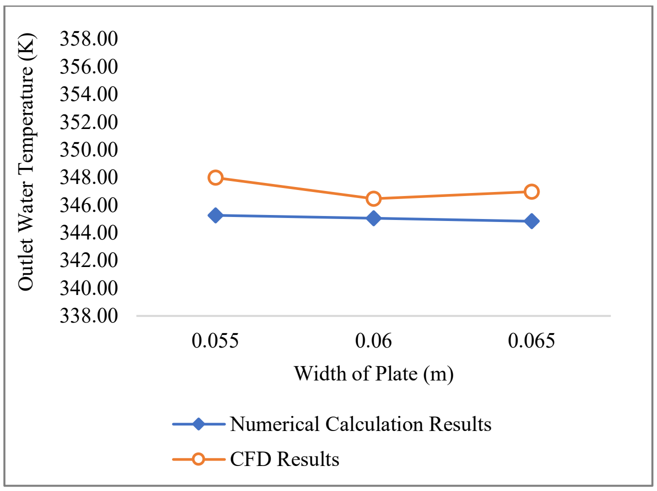


Fig. 24 Effect of change in 'width of plate' at constant 'inlet water velocity' of 0.00625m/s and constant 'inlet air velocity' of 12m/s on 'outlet water temperature'

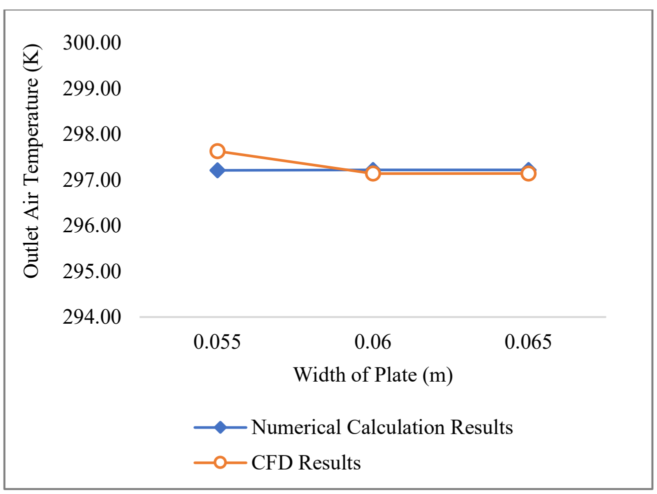


Fig. 25 Effect of change in 'width of plate' at constant 'inlet water velocity' of 0.00625m/s and constant 'inlet air velocity' of 12m/s on 'outlet air temperature'

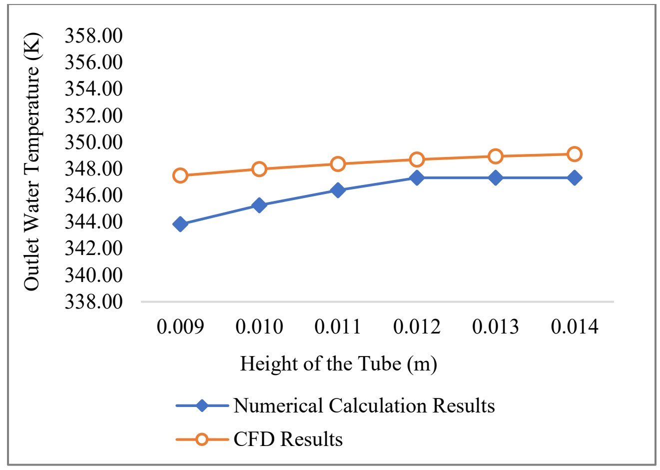


Fig. 26 Effect of change in 'height of tube' at constant 'inlet water velocity' of 0.00625m/s and constant 'inlet air velocity' of 12m/s on 'outlet water temperature'

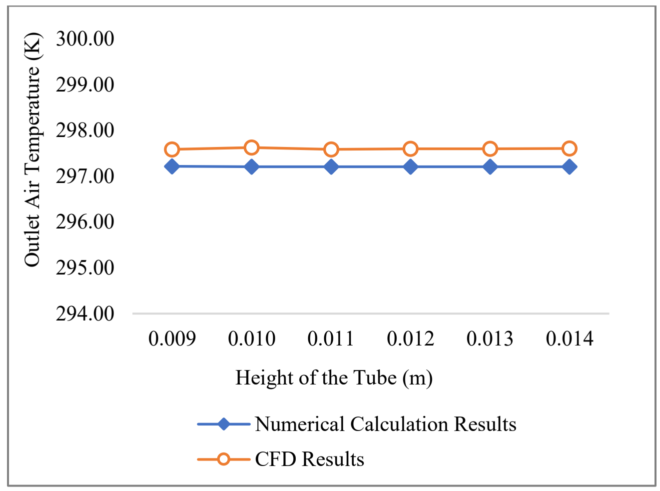


Fig. 27 Effect of change in 'height of tube' at constant 'inlet water velocity' of 0.00625m/s and constant 'inlet air velocity' of 12m/s on 'outlet air temperature'

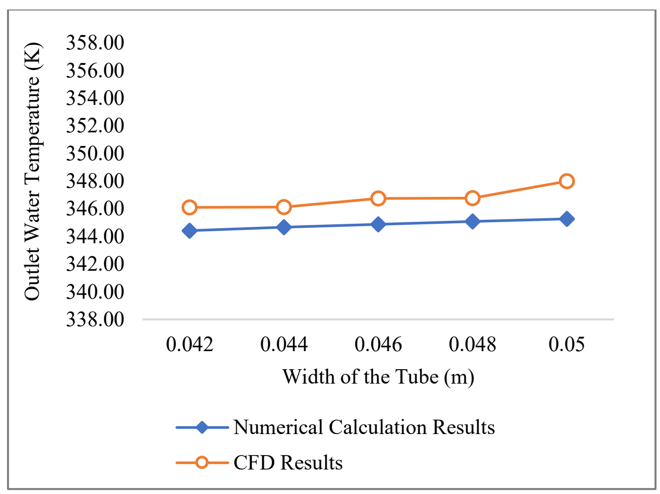


Fig. 28 Effect of change in 'width of tube' at constant 'inlet water velocity' of 0.00625m/s and constant 'inlet air velocity' of 12m/s on 'outlet water temperature'

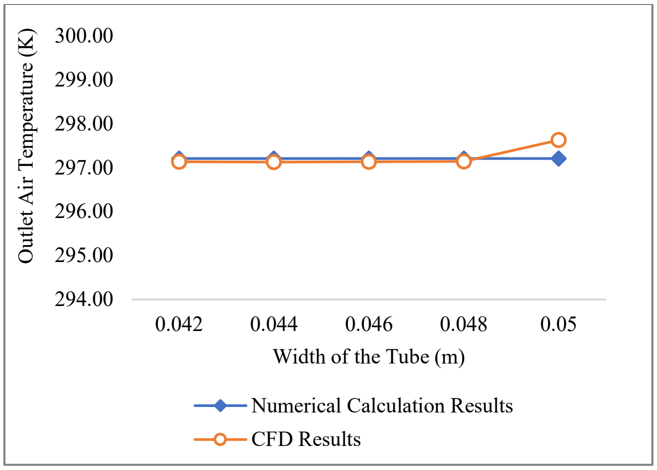


Fig. 29 Effect of change in 'width of tube' at constant 'inlet water velocity' of 0.00625m/s and constant 'inlet air velocity' of 12m/s on 'outlet air temperature'

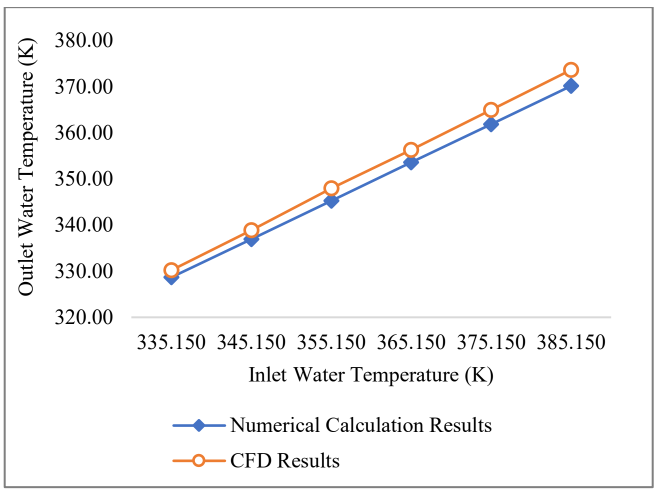


Fig. 30 Effect of change in 'inlet water temperature' at constant 'inlet water velocity' of 0.00625m/s and constant 'inlet air velocity' of 12m/s on 'outlet water temperature'

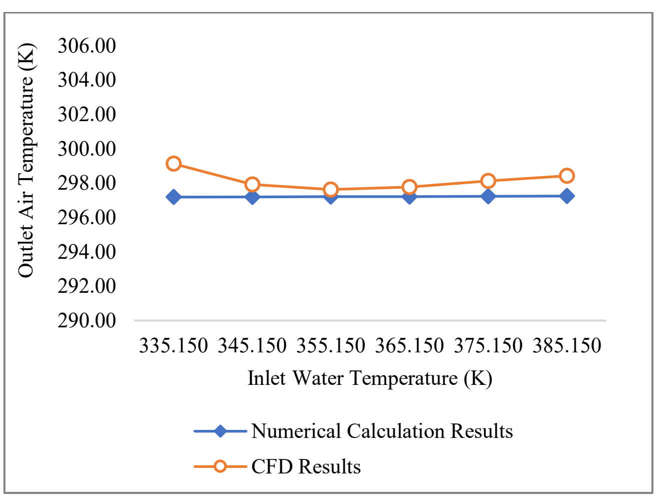


Fig. 31 Effect of change in 'inlet water temperature' at constant 'inlet water velocity' of 0.00625m/s and constant 'inlet air velocity' of 12m/s on 'outlet air temperature'

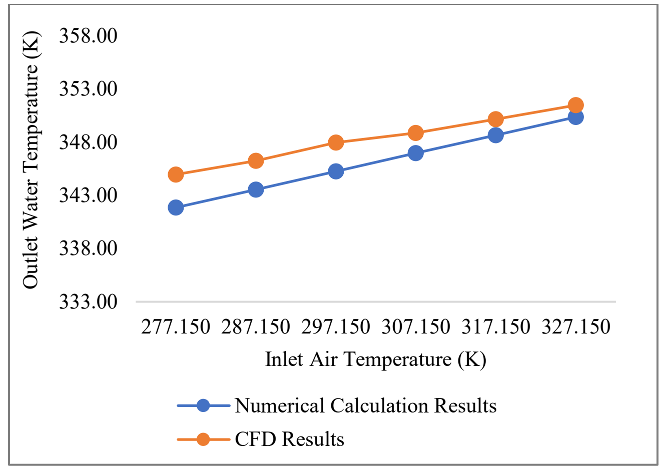


Fig. 32 Effect of change in 'inlet air temperature' at constant 'inlet water velocity' of 0.00625m/s and constant 'inlet air velocity' of 12m/s on 'outlet water temperature'

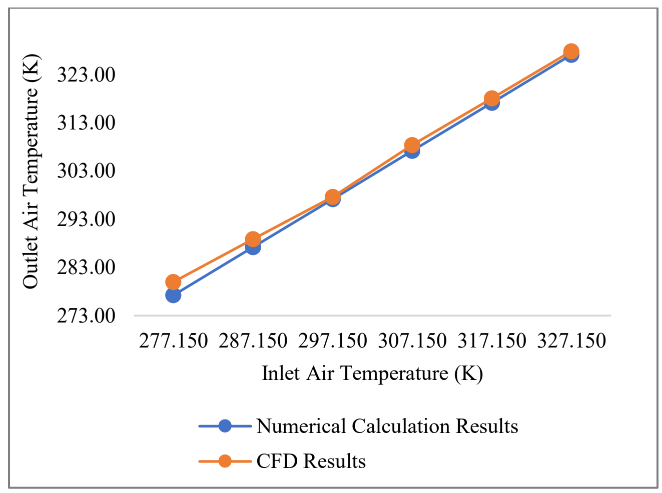


Fig. 33 Effect of change in 'inlet air temperature' at constant 'inlet water velocity' of 0.00625m/s and constant 'inlet air velocity' of 12m/s on 'outlet air temperature'

- Outlet Water temperature predicted using Numerical analysis is accurate upto a minimum of 98.94% when compared to CFD results.
- Outlet Air temperature predicted using Numerical analysis is accurate upto minimum of 99.02% when compared to CFD results
- Combination of the lowest Inlet Water Velocity at 0.0625m/s and Highest Air Velocity at 24m/s shows the maximum reduction in Outlet Water temperature which is at about 10.5K as per CFD results and 11.56K as per numerical calculations.
- For combinations of thickness (across the airflow) and width (along the airflow path) of the plate, the lowest temperature of the water outlet is observed due to the increase in width of the plate
- For combinations of height (across the airflow) and width (along the airflow path) of the tube, the lowest temperature is observed at the minimum height of the

tube. The effect of change in the tube width is there but very minor.

- As the temperature of inlet water is increased, then the difference between inlet and outlet water temperature also increases
- When the temperature of inlet air is at its lowest, then the difference between Inlet and Outlet water temperature is maximum

5. Conclusion

In this study, both numerical calculation and CFD simulation are performed on simple tube and plate-type heat exchangers to analyze their thermal profile and cooling efficiency.

Water Velocity: Numerical and CFD methods indicate that an increase in the inlet water velocity leads to an increase in the outlet water temperature. There is a difference in air temperature values between numerical and CFD simulations, mainly because of the more realistic flow patterns in CFD simulations.

Air Velocity: By increasing the inlet air velocity, it is observed that outlet water temperature decreases; this is prominent at low velocities of water but negligible at high velocities of water. Such heat transfer is mainly affected by the turbulence effects caused due to changes in velocities. Plate

Thickness & Width: When there is an increase in the plate thickness, the frontal area for heat transfer also increases, and Outlet water temperature reduces. However, the outlet air temperature does not change significantly. Plate width has little effect on the results as the flow direction is parallel to the width of the plate.

Tube Dimension: As the tube height and width increase, the outlet water temperature rises in both methods; however, the outlet air temperature does not change appreciably. This implies that geometric changes mainly apply to outlet water temperature.

Effects of inlet fluid temperatures: As the temperature of inlet water increases, the difference between inlet and outlet water temperatures also increases. When the temperature of Inlet Air is at its lowest, then the difference of Inlet and Outlet water temperature is the maximum.

Comparatively, the numerical method is quicker to get the initial results, especially when knowing the internal hot fluid (water) outlet temperature. However, it may need a more accurate formulation to accurately predict the performance of external cold fluid (air). This may be due to the complex behaviour of fluids while interacting with complex shapes and geometries of heat radiator tubes and fins. CFD is better comparatively in terms of accuracy, but it needs large computational facilities, and the time required for simulation is also very high. In summary, the numerical and CFD methods presented above accurately determine the heat exchanger's performance, with minor disparities inherent from the finer detailed flow solution characteristic of the CFD approach. It contributes to reliability and helps enhance the design of the cooling system.

5.1. Future Research Directions

Potential future research directions,

- The above results derived from numerical and CFD analysis can be experimentally validated using an appropriate test setup.
- Numerical methods can be further refined to identify the cooling performance-related parameters accurately.
- The setup conditions of the CFD model can be further refined by adjusting the flow characteristics so that accurate results can be predicted. Also, local area simulation can be explored to reduce computing time and power.
- The impact of other properties, such as material surface roughness, can also be checked in the future, other than the geometry of tubes and plates.

5.2. Practical Application of Findings

- The above findings can help optimize the various parameters of the tube and fins of the radiator, such as width, height, and thickness.
- Also, it is possible to decide on the number of tubes and plates required to achieve desired cooling performance. Knowing the number of tubes, plates, and spacing between them can help to define the maximum size required for the radiator core.

Acknowledgments

The authors are grateful to the Department of Mechanical Engineering of D. Y. Patil College of Engineering, Akurdi, for permitting them to conduct this research. Authors are also grateful to our Research Guide ‘Dr. Pravin T. Nitnaware’ and Co-Guide ‘Mr. Ravikant K. Nanwatkar’ for their valuable suggestions and guidance during the course of the present study.

Also, special thanks to ‘Dr. Premendra J. Bansod’ for their support of this work. I am also thankful for the Student version of Ansys Fluent, which is made available by Ansys Inc. I also appreciate Jetbrains and Project Jupyter for making Pycharm and Jupyter Notebook IDE available for coding Python scripts.

References

[1] Arjun Sastiya, and Devendra Singh Sikarwar, "Enhancement of Radiator Efficiency using CFD Analysis," *International Journal of Research Publication and Reviews*, vol. 4, no. 1, pp. 1306-1311, 2023. [Publisher Link]

[2] R. Paul Linga Prakash et al., "Design and Modification of Radiator in I.C. Engine Cooling System for Maximizing Efficiency and Life," *Indian Journal of Science and Technology*, vol. 9, no. 2, pp. 1-7, 2016. [CrossRef] [Google Scholar] [Publisher Link]

[3] P.S. Amrutkar, and S.R. Patil, "Automotive Radiator Sizing and Rating - Simulation Approach," *IOSR Journal of Mechanical and Civil Engineering*, vol. 2, pp. 1-5, 2013. [Google Scholar] [Publisher Link]

[4] R. Badgujar Pankaj, S. Rangarajan, and S.R. Nagaraja, "Analytical Performance Analysis of Cross Flow Louvered Fin Automobile Radiator," *3rd International Conference on Design, Analysis, Manufacturing and Simulation*, vol. 172, pp. 1-6, 2018. [CrossRef] [Google Scholar] [Publisher Link]

[5] K. Arasu, and T. Ramesh, "Validation of Design Parameters of Radiator Using Computational Tool," *International Research Journal of Engineering and Technology*, vol. 4, no. 11, pp. 825-830, 2017. [Google Scholar] [Publisher Link]

[6] Paul O. Okhiria, and Olumide A. Towoju, "Design and Analysis of an Automobile Radiator with a 14ºC Cooling Capacity," *International Journal of Scientific Research and Engineering Development*, vol. 4, no. 5, pp. 1147-1154, 2021. [Google Scholar] [Publisher Link]

[7] Matthew Carl et al., "The Theoretical and Experimental Investigation of the Heat Transfer Process of an Automobile Radiator," *ASEE Gulf Southwest Annual Conference*, El Paso, Texas, pp. 1-12, 2012. [Google Scholar]

[8] Ernest Palacios Surribas, and Carles Oliet Casasayas, "*Analysis of Automotive Thermal Management Heat Exchangers*," Master's Thesis, Master University in Thermal Engineering, pp. 1-80, 2022. [Google Scholar] [Publisher Link]

[9] Ruchit Doshi et al., "Automobile Radiator Design and Validation," *International Research Journal of Engineering and Technology*, vol. 5, no. 11, pp. 1358-1365, 2018. [Publisher Link]

[10] Utkarsh Wani, Shivam Kulkarni, and Abhishek Keskar, "CFD Analysis of Cross Flow Heat Exchanger and Experimental Validation," *International Journal of Innovative Research in Science, Engineering and Technology*, vol. 8, no. 1, pp. 80-103, 2019. [Google Scholar] [Publisher Link]

[11] Khin Mar Ko, and Mon Mon Zaw, "Design Analysis of Car Radiator in an Engine Cooling System," *International Journal of Research Publications*, vol. 18, no. 1, pp. 1-12, 2018. [Google Scholar] [Publisher Link]

[12] Manichander Rama et al., "Design and Thermal Analysis of an Automotive Radiator for Enhancing Flow Uniformity using CFD," *International Research Journal of Engineering and Technology*, vol. 9, no. 4, pp. 2872-2879, 2022. [Publisher Link]

[13] José Canazas et al., "Heat Transfer and Pressure Drop Performance of a Hydraulic Mining Shovel Radiator by Using Ethylene Glycol/Water-Based Al_2O_3 Nanofluids," *International Journal of Heat and Technology*, vol. 40, no. 1, pp. 273-281, 2022. [CrossRef] [Google Scholar] [Publisher Link]

[14] Ishwar Jagan Dhangar, and M.K. Chopra, "Performance Analysis of Heat Exchanger Using Different Materials by CFD at Different Mass Flow Rate of Water and Air Velocity," *Journal of Emerging Technologies and Innovative Research*, vol. 4, no. 11, pp. 593-600, 2017. [Publisher Link]

[15] P. Jeevananth et al., "Performance Analysis of Automobile Radiator," *International Research Journal of Engineering and Technology*, vol. 6, no. 3, pp. 98-102, 2019. [Google Scholar] [Publisher Link]

[16] Chi-Chuan Wang, Kuan-Yu Chi, and Chun-Jung Chang, "Heat Transfer and Friction Characteristics of Plain Fin-and-Tube Heat Exchangers, Part II: Correlation," *International Journal of Heat and Mass Transfer*, vol. 43, no. 15, pp. 2693-2700, 2000. [CrossRef] [Google Scholar] [Publisher Link]

[17] S. Basavarajappa, G. Manavendra, and S.B. Prakash, "A Review on Performance Study of Finned Tube Heat Exchanger," *Journal of Physics: Conference Series*, Bengaluru, India, vol. 1473, pp. 1-11, 2020. [CrossRef] [Google Scholar] [Publisher Link]

[18] Gandi Saichandar, and Korem Ratna Kumari, "Design and Analysis on Compact Fin Heat Exchangers with Perforations," *International Journal & Magazine of Engineering, Technology, Management and Research*, vol. 3, no. 9, pp. 1229-1236, 2016. [Google Scholar] [Publisher Link]

[19] Akshay P. Shah, and Yogesh D. Vora, "Modification and Design of Finned Tube Heat Exchanger," *Journal of Emerging Technologies and Innovative Research*, vol. 5, no. 5, pp. 700-705, 2018. [Publisher Link]

[20] Lei Zhang et al., "Numerical Study of Fin-and-Tube Heat Exchanger in Low-Pressure Environment: Air-Side Heat Transfer and Frictional Performance, Entropy Generation Analysis, and Model Development," *Entropy*, vol. 24, no. 7, pp. 1-19, 2022. [CrossRef] [Google Scholar] [Publisher Link]

[21] Ali Sabri Abbas, and Ayad Ali Mohammed, "Enhancement of Plate-Fin Heat Exchanger Performance with Aid of (RWP) Vortex Generator," *International Journal of Heat and Technology*, vol. 41, no. 3, pp. 780-788, 2023. [CrossRef] [Google Scholar] [Publisher Link]

[22] M.R. Jafari Nasr, and A.T. Zoghi, "Full Analysis of Low Finned Tube Heat Exchangers," *International Journal of Engineering*, vol. 16, no. 3, pp. 217-232, 2003. [Google Scholar] [Publisher Link]

[23] Y.Q. Wang et al., "Numerical Study on Plate-Fin Heat Exchangers with Plain Fins and Serrated Fins at Low Reynolds Number," *Chemical Engineering and Technology*, vol. 32, no. 8, pp. 1219-1226, 2009. [CrossRef] [Google Scholar] [Publisher Link]

[24] Susan W. Stewart, and Sam V. Shelton, "Design Study Comparison of Plain Finned Versus Louvered Finned-Tube Condenser Heat Exchangers," *Proceedings of the ASME Summer Heat Transfer Conference*, Las Vegas, Nevada, USA, vol. 1, pp. 703-710, 2003. [CrossRef] [Google Scholar] [Publisher Link]

[25] Ishwar J. Dhangar, and Manojkumar Chopra, "Experimental Investigation and CFD Analysis Performance of Fin and Tube Heat Exchanger with Different Types of Fins," *International Journal of Engineering Research & Technology*, vol. 5, no. 1, pp. 1-6, 2017. [CrossRef] [Google Scholar] [Publisher Link]