Wasit Journal of Engineering Sciences

Journal homepage: https://ejuow.uowasit.edu.iq

Effect of the tube material on the thermal performance of automobile (radiator) of cooling system

Ali Abdul Jabbar Ibrahim[1], Zena Khalefa Kadhim[1], and Kamel A. Khalaf[1]

Affiliation
[1]Department of mechanical Engineering,
Wasit University.

Correspondence
Ali Abdul Jabbar Ibrahim

Std2022203.A.Abdul@uowasit.edu.iq

Received
19-April-2024
Revised
16-July-2024
Accepted
22-July-2024

Doi: 10.31185/ejuow.Vol12.Iss3.553

Abstract

The car cooling systems, was considered one of the most important parts which work in dissipated the heat is generating by engine of the car. This study used simulation of CFD to optimize the radiator thermal performance by changing the pipe material while maintaining the same cross-sectional area. By used water-Ethelyn-glycol mixture (50:50) as a hot fluid at flow rates of (12) and (24) litters/hour at different temperatures (85, 90, 95) °C, air was considered a cold fluid with different speeds, respectively (1.5, 2.5, 4.5, 5.5) m /s and its inlet temperature (35) °C . It was clear from the results that there was a significant convergence in heat transfer rates when replacing the pipe metal from copper to aluminium for the improved model compared to the basic model. They are approximately equal at some air velocity. with a small advantage for copper of approximately 5% in terms of heat transfer rate and overall heat transfer coefficient.

Keywords: heat exchanger, CFD simulation, thermal performance, heat transfer.

الخلاصة: تعزيز التبادل الحراري عددياً في المبادل الحراري المتكامل الذي يستخدم في أنظمة تبريد السيارة والذي يعتبر من أهم أجزاء السيارة في استخدمت الدراسة محاكاة نشر الحرارة المتولدة من محرك السيارة لتقديم تصميم جديد للراديتر يتضمن تغيير مادة الأنابيب. وهذا ما تضمنته الدراسة مع الحفاظ على نفس مساحة المقطع العرضي. استخدمت الدراسة خليط الماء- ايثيلين جلايكول (50:50) كسوائل ساخنة بمعدل تدفق (24,12) لتر/ساعة، وفي درجات حرارة مختلفة °C (85، 90، 95)، واعتبر الهواء سائل بارد مع سرعات مختلفة على التوالي (1.5، 2.5، 4.5، 5.5) م/ث ودرجة حرارة مدخلها (35) درجة مئوية. اتضح من النتائج التي ظهرت أن هناك تقارب كبير في معدلات انتقال الحرارة عند استبدال معدن الأنابيب من النحاس إلى الألومنيوم للنموذج المصمم مقارنة بالنموذج الأساسي وهي متساوية تقريبا عند بعض سرعات الهواء.

1. INTRODUCTION

The device which is the compact heat exchanger for transferring the heat through two liquids of different temperatures [1]. Which is used in many different industrial fields such as power plants, automobile radiators, air conditioning and refrigeration, etc.[2]. Removing heat from your vehicle's engine is essential to achieving optimal system performance [3] .This work includes a study of the compact heat exchanger. Especially those consisting of oval tubes and ventilation fins. By numerical simulation, to obtain the thermo-hydraulic behaviour in the fin geometry of the ventilation pipe. The study included typical entry speeds corresponding to low Reynold numbers. Until the behaviour of these geometric shapes is simulated. Some models have shown an increase of up to 200% when compared to plate fins [4] .

This study involves finding a new car radiator design that is more compact while achieving higher levels of heat transfer. This is done by comparing a regular straight tube radiator to a new spiral tube radiator. Modelling was performed using SolidWorks, while fluid flow analysis was performed using Ansys Fluent [5] . A theoretical performance analysis is conducted for different heat exchanger models, including the counterflow radiator, the crossflow radiator, and a group of the heat exchangers used in a single unit vehicle radiator, Cross and Counter Flow heat exchanger called CCFC, have been optimized in performance. Current analysis shows that the radiator has 35.61% and 27.44% better cooling capacity compared to conventional crossflow radiators at vehicle speeds of 40 km/h and 80 km/h respectively with a mass water flow rate of up to 2 kg/s [6] .

Computational Fluid Dynamics (CFD) software was using for studying the effect of air inlet temperature, fluid velocity to optimize the efficiency of the vehicle radiator and find the ideal geometry. The comparison of the drop of pressure and transfer of heat of the heat exchanger with different parameters to obtain the best performance. Various results show that CFDs have proven to be very effective in reducing cost and time from concept to production [7] .

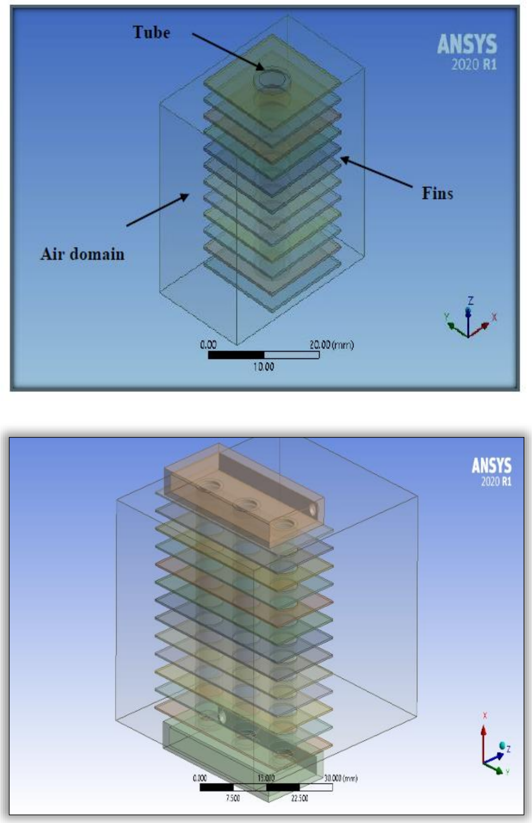
Current study deal with the improving the heat production of the radiator's ventilation, by changing the longitudinal vertical convection fins distribution. The radiator's ventilation, combined the supply of air's ventilation with emissive of heat into room, had a higher force of air through radiator's plates when comparing to conventional radiator [8] .

The structure of the field's flow and the cool medium of the tube presented, which has a significant impact on the heat transfer rates of automobile radiators. Two new types of cooling tubes were developed ″Wasp waisted tube 2 and Wasp-waisted Tube 3, the first having a larger hydraulic diameter than the second and having equal cross-sectional area for both tubes. Simulations were performed for six types of cooling pipes with different structure flow field and numerically equal cross-sectional flow area. The results proved that the transfer of heat capacity of the ″wasp tube 2 and the wasp tube 3″ was increasing by 10.6% and 3.5% respectively was much better than that of other radiator tubes [9] .

Numerical simulations were performed between two exchangers, one with fin and aluminium tube and the second consist of copper tube and aluminium fins. The results were compared with regard to heat transfer performance of heat transfer and cost were as follows. Because the conductivity of copper is high, the performance of a heat exchanger with copper tube is higher than another consisting of aluminium tubes, depending on the speed of the incoming air. Heat transfer for radiator which consist of aluminium tubes is (4%-12%) less than for one with copper tubes of the same structure, and the coefficient of heat transfer is 7%-9% lower. The cost of radiator with aluminium tube and fin is only 8% because of the low density and cheaper than copper. Opposite the cost of the one copper tube and aluminium fins [10] .A numerical investigation was carried out to examine how tube shape impacts the transfer of heat and the flow of fluid in a tube finned heat exchanger, utilizing Ansys Fluent software. Various shapes (circular, flat, oval, and elliptical in both left and right directions) were assessed, with simulations conducted for two-dimensional external flow of an incompressible fluid within Reynolds numbers ranging from 3000 to 20000. The findings reveal that tube shape directly influences the thermal and dynamic performance of the heat exchanger. Specifically, the circular tube demonstrates an approximately 18% higher heat transfer coefficient compared to the flat tube and incurs an average pressure drop around 10% lower under identical conditions. Conversely, the oval tube exhibits the least pressure losses relative to other configurations. Relative to the flat tube, the RO, oval, LO, and elliptical tubes ensure heat transfer coefficients approximately 18%, 12%, 10%, and 4% higher, respectively [11]. explain the effect of improving the structure of the tube finned unit radiator on heat transfer and the flow of fluid. This is what the researcher did in this paper. An aluminium serrated fin structure was used as an alternative to the copper fin while keeping the size and position of the flat copper tube with hot water was unchanging. Under variables such as atmospheric pressure of 85040 Pa, the temperature of ambient is 24.6°C, and the density of air is $\rho = 0.94\ kg/m^3$, the analysis of simulation CFD was performed by ″ICEPAK17.0″. To compare changes in the turbulence field, velocity field, and temperature field. This increased fluid turbulence is the benefit of the toothed new heat sink structure, as well as reducing the outer boundary layer's thickness of the pipe's water, finally enhancing the effect of heat transfer of the coolant. The result showed that heat spread of the serrated aluminium plates is greater than that of copper plates, and the coefficient of heat transfer increases by about 1.3%, the average pressure decreases, and the turbulence performance is enhanced [12]. presented the study of the effect of various factors, including orifice arrangement, on rheology, the drop of pressure, and the transfer of heat in compact fin-and the tube heat exchangers study by using the ε-NTU method. Parameters such as pitch variation, angle, number of openings, flow redirection within the fin series, as well as the lengths of the unvented entry and exit fins were analysed. The results indicate that streamline reorientation depends on the vent angle and the distance of the redirection site from the fin inlet, and not on exposed fin lengths or the number of vents. However, aligning the suction or blowing directions of the multiple fins resulted in a significant 40% increase in heat transfer [13] . This study aims to experimentally and numerically explore the feasibility of using ″pulsating heat pipes (PHPs)″ as alternatives to fin in a standard cool air heat exchanger. Considering the minimum of the difference of temperature through cool air and the indoor flow of air in the study case, it is necessary to study suitable ″PHPs″ capable of operating under small temperature differences and choose the appropriate of the fluid working. Next, test rig was designed and built. Computational fluid dynamics was used for predicting the behaviours of main tube equipped with pulsate heat pipe and finned tube. Results showed a significant improvement in heat transfer when using pulsated heat pipe as fins. In free convection (no external flow through the tubes), the heat pipes outperformed the fins, resulting in a significant 310% improvement in overall coefficient of the heat transfer in the best case. This improving was about 263% under forcing load conditions [14]. This research includes the use of a circular cross-section fan. The results showed that using it for the radiator gives the maximum temperature drop and the minimum pressure drop for the different cases used to study radiator performance. Using the fan and changing its shape enhances engine cooling rates, thus reducing the size of the cooling system by improving heat removal from the engine, lightweight and small radiators. All of these improvements lead to increased fuel economy[15]. It was found that fins with ventilation holes improve heat dissipation due to the larger fin area and the resulting turbulence in the air due to the holes. After conducting theoretical analysis of the results using CFD, an overall increase of 55% was found in the heat transfer rate. It was also found that the ventilated fin model led to a reduction in the size of the radiator by 43.33%, while reducing the weight by only 6.27%. The structural study demonstrates that the vented fin model is structurally valid and can withstand the actual working conditions of the radiators. Further studies can be conducted by changing the size of the ventilation holes, the angle of the ventilation holes, and the ventilation holes to choose the best one [16].

2. THE MATHEMATICAL MODEL AND BOUNDARY CONDITIONS

Two models of CFD; the basic and the enhanced models that were using to resolve the thermal achievement for small-scale cross-flow heat exchanger. Table (1) shows two heat exchangers with the dimensions. geometries were creating by use ″ANSYS FLUENT″ software (2020 R.1), and consist of 3 parts: the air field, the tube, and the water inside the tube, which are geometrically installed in one section. Figure 1 shows the geometry of the two models. The flow is laminar in hot fluid occurs at temperatures (85, 90, 95) ºC and a mass flow of (12,24)L/h. Table″ 2″ shows the physic thermal properties for hot water[17]. The air was cooling in a speed of (1.5, 2.5, 4.5, 5.5) m/s and at a temperature of (35) °C.



a b

Fig.1 Geometry for both models. (a) Base model (b) Enhancement model

Table 1 The Heat Exchangers with Dimension of it

Types of Heat exchanger	Shape of tubes	Material of tubes	Shape of fins	Length of tube (m)	Internal diameter (m)	External diameter (m)	Cross sectional area of the tube (m2)
Base model	Circular	copper	Plain	0,34	0,0047	0.0063	$1,74*10^{-5}$
enhancement model	Circular	Aluminium	Plain	0,34	0,0047	0.0063	$1,74*10^{-5}$

Table 2 the physic thermal properties for hot Fluid [17] .

Density (kg/m3)	The thermal of conductivity (w/m. °C)	Specific heat (J/kg.k)	Dynamic viscosity (kg/m.s)
998.2	0.6	4182	0.001003

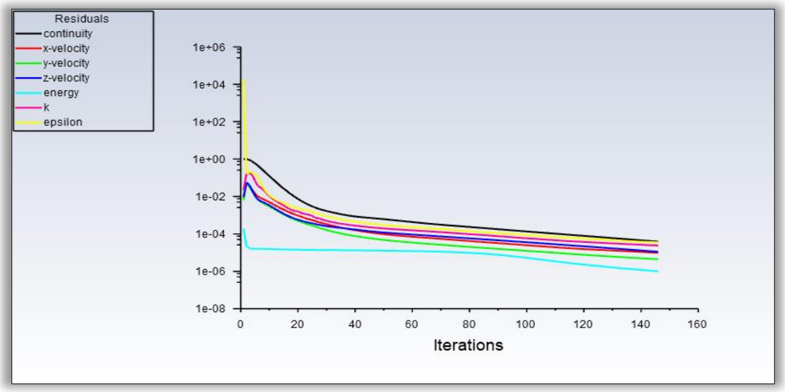


Fig. (2) The Simulation Prosses.

3.GOVRRNING EQUATION

The solution of the governing equation is given and the simulation process is completed under the assumption is utilized in the simulation. These assumptions involve flow under steady-state conditions, constant physical properties with temperature, without heat loss, heat generation, phase change, and with neglect of radiation effects and without mixing processes [18] .

- Mass conservation ″continuity equation″:

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

- ″Conservation of Momentum″:

- **x-axis**

$$″\bar{u}\frac{\partial \bar{u}}{\partial x}+\bar{v}\frac{\partial \bar{u}}{\partial y}+\bar{w}\frac{\partial \bar{u}}{\partial z}"+\left("\frac{\partial}{\partial x}(\acute{u}^2)+\frac{\partial}{\partial y}(\acute{u}\bar{v})+\frac{\partial}{\partial z}(\acute{u}\bar{w})"\right)=\frac{1}{\rho}\frac{\partial p}{\partial x}+\frac{\mu}{\rho}\left("\frac{\partial^2 \bar{u}}{\partial x^2}+\frac{\partial^2 \bar{u}}{\partial y^2}+\frac{\partial^2 \bar{u}}{\partial z^2}"\right)$$

- **y-axis**

$$(\bar{u}\frac{\partial \bar{v}}{\partial x}+\bar{v}\frac{\partial \bar{v}}{\partial y}+\bar{w}\frac{\partial \bar{v}}{\partial z})+\left("\frac{\partial}{\partial x}(u\acute{\bar{v}})+\frac{\partial}{\partial y}(\bar{v}^2)+\frac{\partial}{\partial z}(\bar{v}\bar{w})"\right)=\frac{1}{\rho}\frac{\partial p}{\partial y}+\frac{\mu}{\rho}"\frac{\partial^2 \bar{v}}{\partial x^2}+\frac{\partial^2 \bar{v}}{\partial y^2}+\frac{\partial^2 \bar{v}}{\partial z^2} \qquad 3$$

- **z-axis**

$$(\bar{u}\frac{\partial \bar{w}}{\partial x}+\bar{v}\frac{\partial \bar{w}}{\partial y}+\bar{w}\frac{\partial \bar{w}}{\partial z})+\left(\frac{\partial}{\partial x}(u\acute{\bar{w}})+\frac{\partial}{\partial y}(\bar{v}\bar{w})+\frac{\partial}{\partial z}(\bar{w}^2)\right)=\frac{1}{\rho}\frac{\partial p}{\partial z}+\frac{\mu}{\rho}(\frac{\partial^2 \bar{w}}{\partial x^2}+\frac{\partial^2 \bar{w}}{\partial y^2}+\frac{\partial^2 \bar{w}}{\partial z^2}) \qquad 4$$

- Conservation of Energy

$$(\bar{u}\frac{\partial T}{\partial x}+\bar{v}\frac{\partial T}{\partial y}+\bar{w}\frac{\partial T}{\partial z})+\left(\frac{\partial}{\partial x}\left(\bar{u}\acute{T}\right)+\frac{\partial}{\partial y}\left(\bar{v}\acute{T}\right)+\frac{\partial}{\partial z}\left(\bar{w}\acute{T}\right)\right)=\alpha(\frac{\partial^2 \bar{T}}{\partial x^2}+\frac{\partial^2 \bar{T}}{\partial y^2}+\frac{\partial^2 \bar{T}}{\partial z^2}) \qquad 5$$

4. GRID INDEPENDENCY AND MESH GENERATION

Mesh generation represents a crucial step in simulations, as it transforms geometry to mini cells (control volumes) to enhance the program’s capability in solving partial differential equations.

Meshes can be categorized into arrangement and unarrangement types. In the research, was chosen tetrahedral mesh due to its efficiency in handling complex geometries. Figure 3 illustrates the computational mesh used in the investigation. The quality of the tetrahedral mesh was evaluated using the Skewness method, with a Skewness fine adjustment of 1 indicating acceptable reliability[19] . Details regarding the number of nodes, elements, and skewness are provided in Table 3. Inlet boundary conditions for this study were determined based on 6 unlike numbers of items, as depicted in Figure 4.

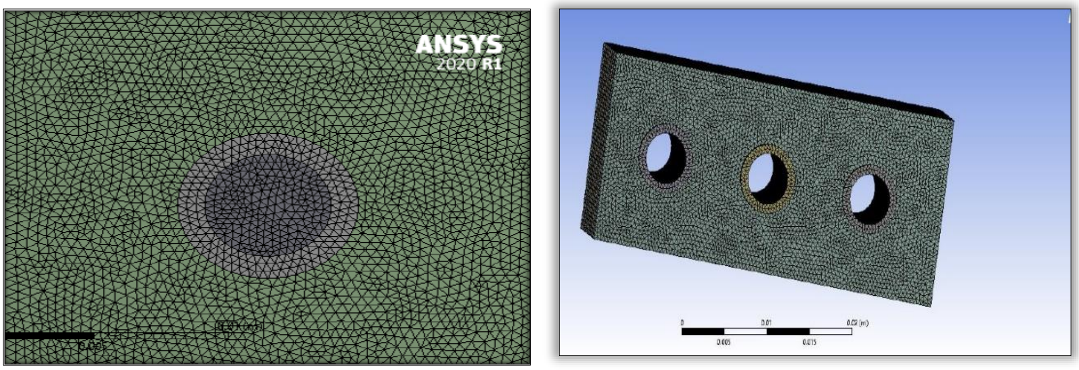


Fig. ″3″ A mesh consisting of the two models. (a) Base model. (b) enhancement model.

Table ″3″: No of nodes, elements and skewness.

Types of H.E	No. of nodes	No. of elements	Skewness
Baseline model	387531	2248512	0.84771
enhancement model	542039	2514322	0.89434

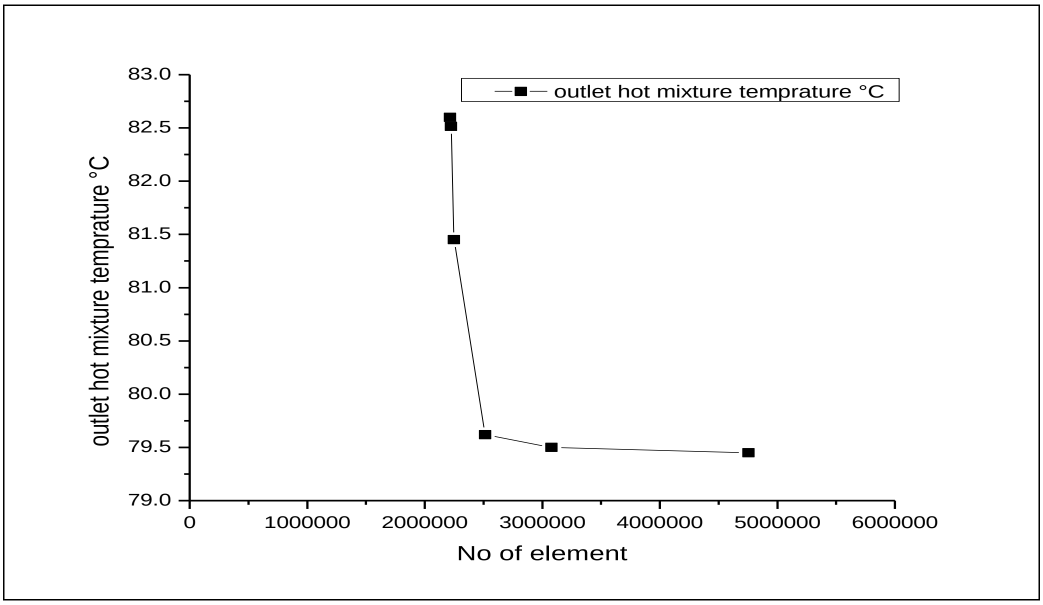


Fig. 4: Grid independent test between number of elements and mixture water outlet temperature (°C)

5. RESULTS AND DISCUSSION

5.1 VALIDATION WORK

The ANSYS Fluent validation case (20R1) was used to verify the accuracy of the program in [1]. The following was used in the previous study: a mixture of water - ethylene glycol (50 - 50) as the hot fluid at four flows (10, 12, 18, 24) litters/hour, and the hot fluid entry temperature was (75, 85, 95), and air was considered as the fluid. Cold, at four speeds (1.5, 2.5, 4.5, 5.5) m/s and at inlet temperature (35). Using circular copper metal tubes with a hydraulic diameter of (0.0047) m and a length of (0.34) m. The results of one of the tubes used were approved, and through the results obtained, it appeared that there was a large match with the results of the previous study, with an error rate of (0.497%).

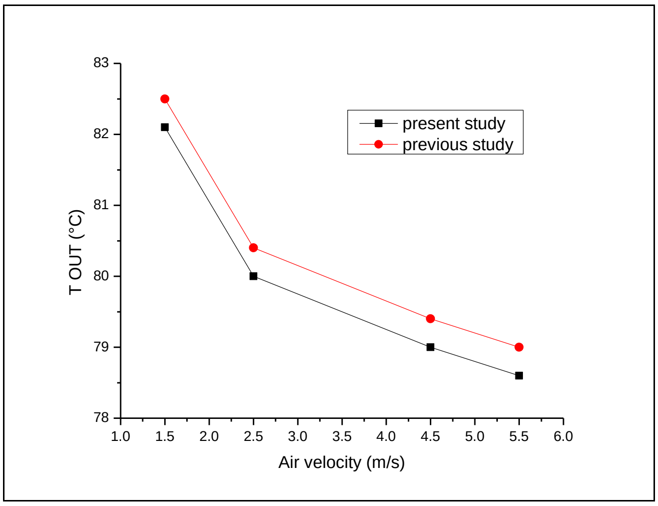


Fig. 5 Validation between the previous[1] and current study of the fluid temperature at the exit and at a flow of 24 L/h.

5.2. HEAT TRANSFER RATE

Figure 5 shows the heat transfer rates among the basic model at a circular tube an internal diameter of 4.7mm, a flow rate of liquid (24) L/h and an inlet temperature of 85°C compared with the enhancement model with the same diameter, under the same boundary conditions, differing only in the metal of the tube. Where copper is for the basic model and aluminium for the enhancement. It has been observed that there is a significant convergence in the heat transfer rate, with a slight relative preference for copper, and it is equal at some air speeds. This due to high thermal conductivity for copper and high specific heat for aluminium’s.

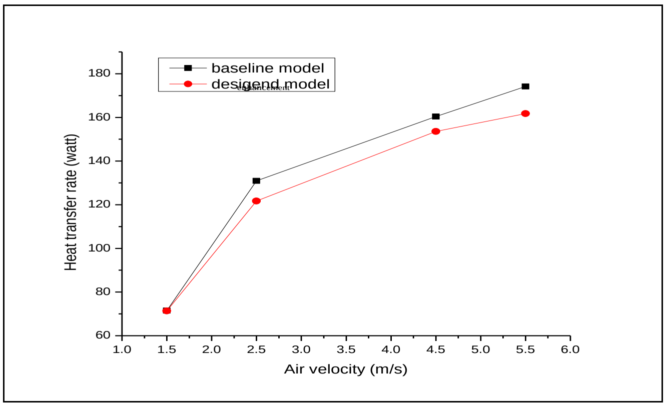


Fig. 6 Show heat transfer rates for base and enhancement models at 24L/h, hot liquid mass flow rate and 85 °C inlet temperature.

5.3: THE OVERALL HEAT TRANSFER COEFFICIENT (UI) IN INNER SIDE

Figure 6 illustrates the impact of air velocity and water inlet temperature on the overall heat transfer coefficients (Ui) of both the baseline and enhancement models on the water side. The findings indicate that (Ui) increases with air velocity for a same behaviour. There is a convergence in the results between the copper tubes of the basic model and the aluminium tubes of the enhancement model.

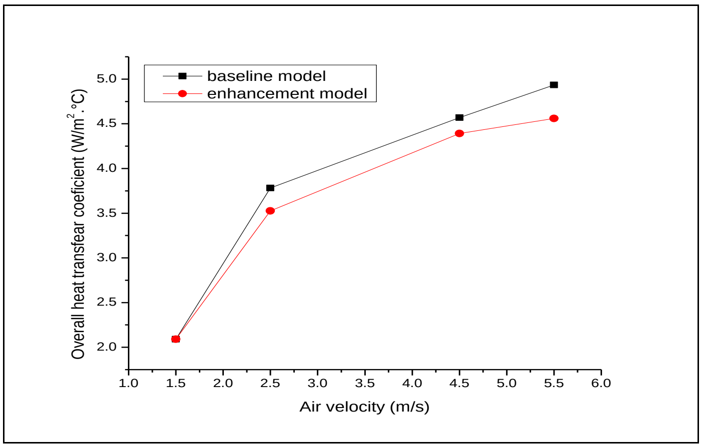
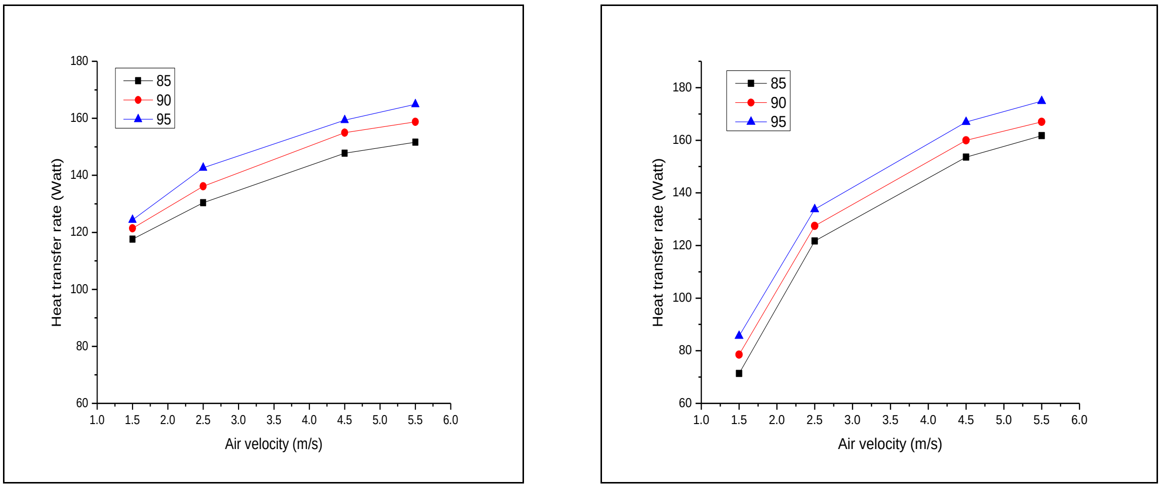


Fig. 7 The overall heat transfer coefficient in Inner side for base and enhancement model at 24 L/h hot liquid mass flow rate and 85 °C inlet temperature.

5.4. IMPACT OF AIR VELOCITY ON HEAT TRANSFER RATE

The impact of different air velocity on the rates of heat transfer to water at different temperatures entering the cooling system (radiator), with two different flows, as shown in Figure (7). The results showed that temperature rates increase with increasing air velocities and water temperatures and increasing flow, due to increased mass and the penetration of high air velocities into the thermal boundary layer more effective.

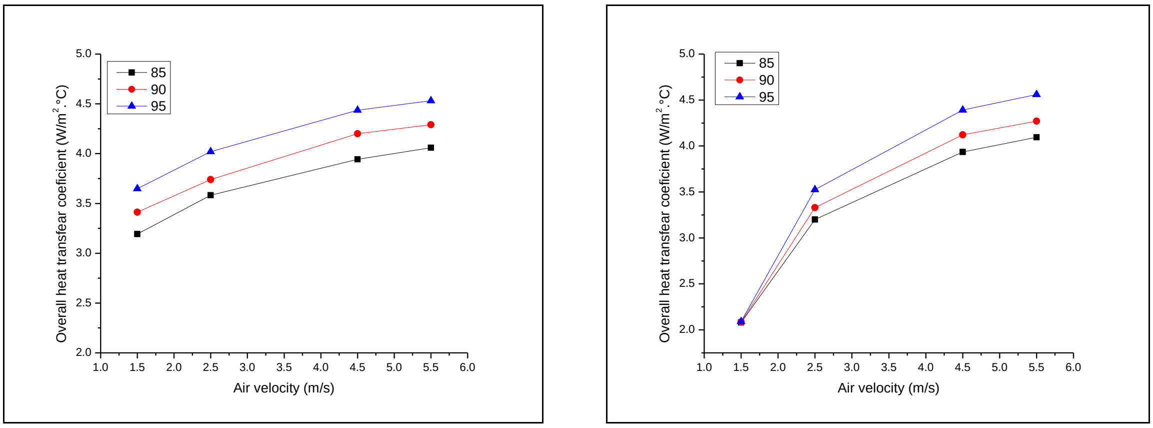


(a) 12 L/h (b) 24 L/h

Fig. (8) Evident the rate of heat transfer versus the air velocity at different inlet water temperature.

5.5. EFFECT OF AIR VELOCITY ON OVERALL HEAT TRANSFER COEFFICIENT (UI).

The relationship between different air velocities and the total heat transfer coefficient at different inlet water temperatures with two mass flow rates seeming in figure (8). The results showed that the total heat transfer coefficient increases with increasing inlet water temperatures, with a relatively small effect of mass flow rate.



(a) 12 L/h (b) 24 L/h

Fig. (9) Obvious the overall heat transfer coefficient versus air velocity with different inlet temperature.

5.6 TEMPERATURE CONTOUR

with hot liquid mass flow rates of 24 L/h and 85°C, and an air velocity of 5.5 m/s. The colours red and blue correspond to high and low temperatures, gradually. Fins are utilized to absorb heat from the tube's wall via conduction and convection along its length. The enhancement and basic model include circular tubes to control the movement of water inside them, and the only difference in the metal of the tubes is copper for the basic and aluminium for the enhancement. Figures 10 and 11 show the temperature gradient on the tube for both models, and indicate an effective heat exchange between the hot mixture and the tube surface and a convergence in temperatures. Figure 12 and 13, Clear the temperatures distribution on walls of the finned tube.

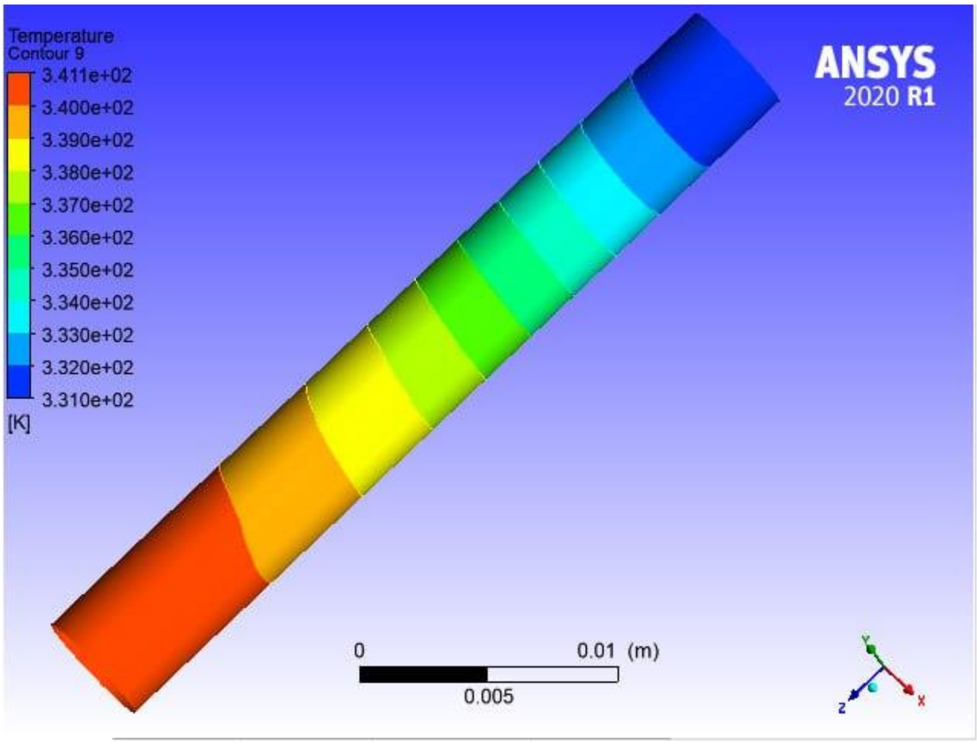


Fig. (10) Temperature division of the tube for base model.

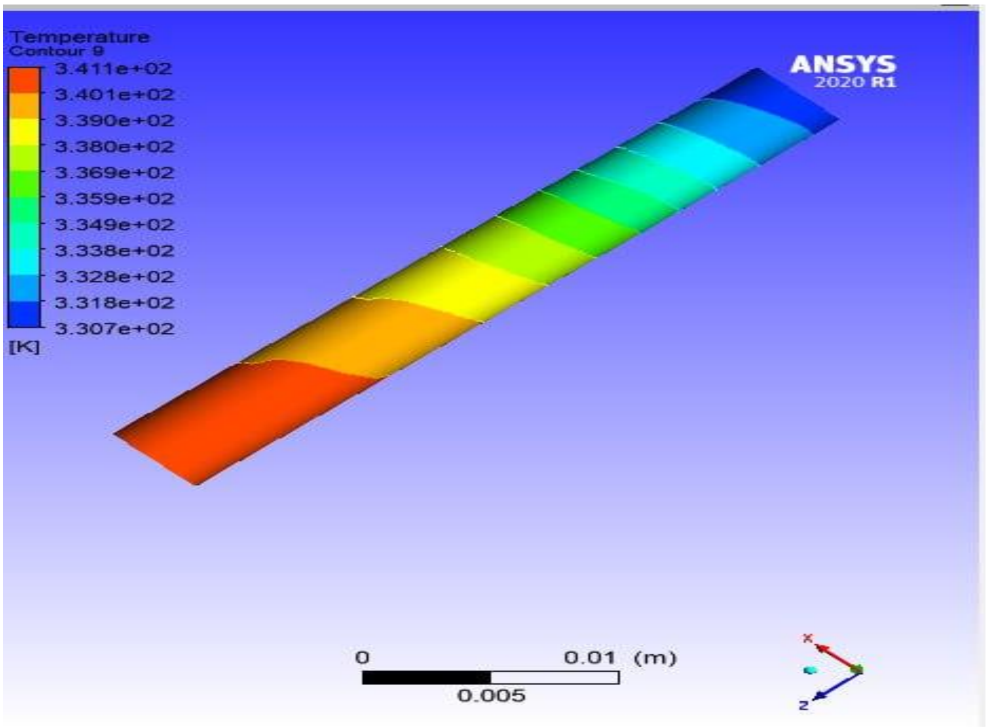


Fig. (11) Temperature division of the tube for enhancement model.

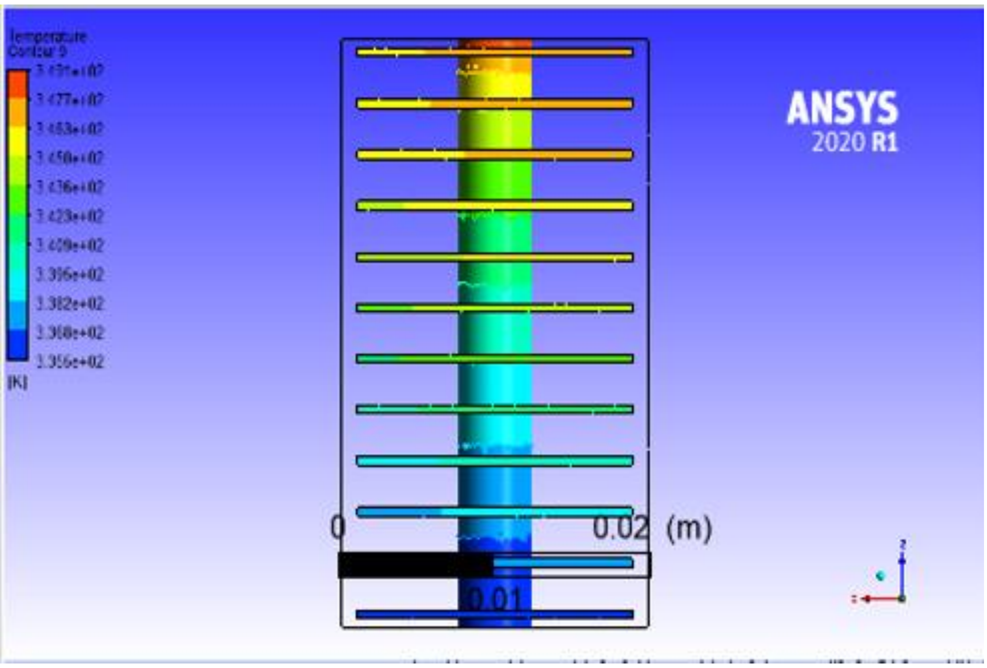


Fig. (12) Temperature division of the tube and fin for bassline model.

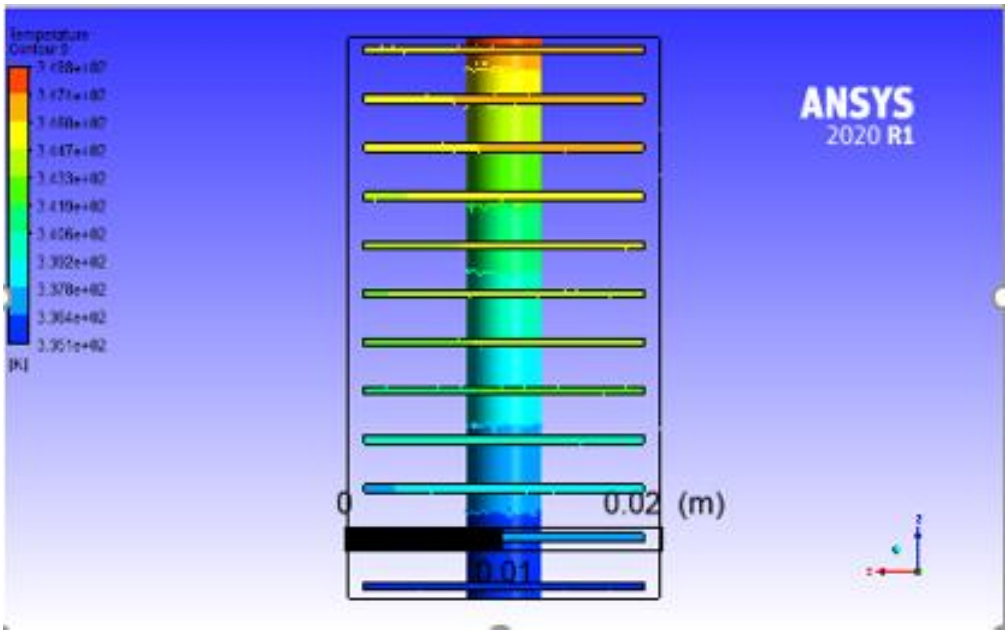


Fig. (13) Temperature division of the tube and fin for enhancement model.

6. CONCLUSIONS

In this study, CFD simulation was used to analyse the thermal performance of a basic compact HE comprising a circular tube and regular fins with identical dimensions and the tube metal being copper. With another heat exchanger, it has the same dimensions, cross-sectional area, and geometric shape, but the tube metal is aluminium. Numerical analysis showed that replacing the tube material from copper to aluminium led to a convergence of values in lowering the temperature, despite the high thermal conductivity of copper compared to aluminium. This is due to the high heat capacity of aluminium. Thus, there is a convergence in the results of the numerical analysis with a small advantage for copper of approximately 5% in terms of the heat transfer rate and the overall heat transfer coefficient.

The values of the total heat transfer coefficient and the amount of dissipated thermal energy in the improved model using aluminium tubes are close to the values of copper tubes under the same conditions. It is recommended to use aluminium tubes in the car's cooling system because they give lighter weight and lower cost compared to the model that contains copper tubes.

REFERENCES

[1] O. A.Najman, “Numerical investigation on Enhancing Heating performance in Automotive Radiator.” p. 9, 2022.

[2] A. Karimi and M. Afrand, “Numerical study on thermal performance of an air-cooled heat exchanger: Effects of hybrid nanofluid, pipe arrangement and cross section.” p. 14, 2018.

[3] A. S. Tijani and A. S. bin Sudirman, “Thermos-physical properties and heat transfer characteristics of water/ anti-freezing and Al2O3/CuO based nanofluid as a coolant for car radiator.” p. 10, 2018.

[4] A. Menéndez-Pérez, M. Textor, L. Pita-Cantos, and R. Borrajo-Pérez, “Determination of the optimum louver angle of a louvered fin with elliptical tubes.” p. 8, 2019.

[5] R. Sathyan, “Analysis of Automobile Radiator Using Computational Fluid Dynamics.” p. 5, 2016.

[6] S. Singh and R. R. Sahoo, “Performance Enhancement in Automotive Radiators with Different Configurations,” *Journal of Aeronautical and Automotive Engineering (JAAE)*, vol. 3, no. 1. pp. 17–22, 2016. [Online]. Available: http://www.krishisanskriti.org/Publication.html

[7] H. B. Patel and M. D. Dinesan, “OPTIMIZATION_AND_PERFORMANCE_ANALYSIS_OF.pdf.” p. 4, 2014.

[8] J. A. Myhren and S. H. Department, “Improving the thermal performance of ventilation radiators e The role of internal convection fin.” p. 9, 2011.

[9] Z. Ma, Y. Huang, X. Chen, J. Song, X. Zhang, and T. Li, “NUMERICAL INVESTIGATION OF EFFECT OF THE FLOW FIELD STRUCTURE AND,” pp. 1–16, 2023.

[10] F. Wang *et al.*, “Comprehensive Evaluation of the Performances of Heat Exchangers with Aluminum and Copper Finned Tubes,” *Int. J. Chem. Eng.*, vol. 2023, 2023, doi: 10.1155/2023/6666947.

[11] D. Sahel, H. Ameur, and M. Mellal, “Effect of tube shape on the performance of a fin and tube heat exchanger,” *J. Mech. Eng. Sci.*, vol. 14, no. 2, pp. 6709–6718, 2020, doi: 10.15282/JMES.14.2.2020.13.0525.

[12] Y. Guan, H. Cui, and J. Fei, “Study on Optimization of Copper to Aluminum for Locomotive Finned Tube Radiator,” *Study Optim. Copp. to Alum. Locomot. Finned Tube Radiator*, pp. 1–14, 2023.

[13] R. Moosavi, A. Vaisi, and K. Javaherdeh, “Investigation of the geometrical structure of louvered fins in fin-tube heat exchangers for determining the minimum distance of the headers,” no. March, p. 12, 2021, doi: 10.1007/s12206-021-0335-4.

[14] H. J. Mosleh, M. A. Bijarchi, and M. B. Shafii, “Experimental and numerical investigation of using pulsating heat pipes instead of fins in air-cooled heat exchangers.” 2019.

[15] C. I. Priyadarsini, P. A. Devi, and T. R. Reddy, “Design and Performance Analysis of Automotive Radiator using Computational Fluid Dynamics.” p. 6, 2022.

[16] S. Bhattarai*, K. D. Parag Vichare, S. K. Ghimire, and D. K. M. Raju, “Computational Fluid Dynamics based Performance Evaluation of Louvered Fin Radiator.” p. 6, 2017.

[17] O. Abdul and H. Najman, “Enhancement of Heat Transfer for New Design Automotive Radiators with Using Nanofluid (CuO),” 2022.

[18] F. A. Taher and Z. K. Kadhim, “Experimental and Numerical Investigation of Different Finned Tube Heat Exchanger Supervised by,” no. January, 2019.

[19] S. A. Mardan and Z. K. Kadhim, “Numerical and Experimental Study of Cross-Flow External Finned Tubes Heat Exchanger,” 2021.