Wasit Journal of Engineering Sciences

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

Numerical investigation on Enhancing Heating performance in Automotive Radiator

Ola A.Najman[1], Zina K.Khadhim[1], Kamel A.Khalaf[1]

Affiliations
[1]Department of Mechanical Engineering,
Wasit University,
Wasit, Iraq.

Correspondence
Zina K.Khadim
drzena@uowasit.edu.iq

Received
29-September-2022
Revised
20-November-2022
Accepted
20-November-2022
Doi: 10.31185/ejuow.Vol10.Iss3.384

Abstract

In this paper, the improvement of heat exchange in a compact heat exchanger that is used in the automotive cooling system is numerically studied due to the importance that this part of the automotive represents in removing the heat generated in the automotive engine. The study adopted CFD simulation to introduce a new design of radiator that included changing the geometry and material of the tube while maintaining the same cross-section area In addition, the study also included replacing the shape of the fins from the louver to plain fin with the increase in the number of tubes. The study used a water-Ethylene glycol mixture (50:50) as a hot fluid with four flow rates (10, 12, 18, 24) l/h, (75, 85, 95) °C inlet temperature, and air as a cold fluid with 1.5, 2.5, 4.5, 5.5) m/s and (35) °C inlet temperature. Through the results obtained, the designed model appeared to have a higher thermal performance than the standard model due to the material and shape of the tube and fins. The designed model performed well, achieving an improvement ratio of (18.6%) for the heat transfer rate, (13.8%) for the overall heat transfer coefficient for the hot fluid, and (29.8%) for the air side heat transfer coefficient

Keywords: Automotive radiator, Compact heat exchanger, CFD simulation, Designed geometry, Heat transfer rate

الخلاصة:في هذا البحث يتم دراسة تحسين التبادل الحراري في مبادل حراري مضغوط يستخدم في تبريد السيارات نظرا للأهمية التي يمثلها هذا الجزء من اجل التخلص من الحرارة المتولدة في محرك السيارة. البحث يقدم دراسة نظريه اعتمدت تغيير شكل الانابيب المستخدمة من مسطحة الى دائريه المقطع مع الحفاظ على نفس المساحة، اضافة الى تغيير معدن الانبوب من المنيوم الى النحاس، الدراسة ايضا شملت تغيير شكل الزعانف الى مستويه مع زيادة عدد الانابيب. تضمنت الدراسة استخدام خليط الماء - الإيثيلين جلايكول (٥٠-٥٠) كسائل ساخن بأربع معدلات تدفق (١٠، ١٢، ١٨، ٢٤) لتر / ساعة، (٧٥,٨٥,٩٥) م ° والهواء كمائع بارد بسرعه (٥,٥) م / ث ودرجة حرارة ٣٥ م°. من خلال النتائج التي تم الحصول عليها النموذج المصمم أظهر أداء حراري مرتفع عن النموذج القياسي بسبب مادة وشكل الأنبوب والزعانف. تم الحصول على نتيجة جيدة للنموذج المصمم مع نسبة تحسن (١٨,٦٪) لمعدل انتقال الحرارة و (١٣,٨٪) لمعامل انتقال الحرارة الكلي للمائع الساخن و (٢٩,٨٪) لمعامل انتقال الحرارة بالحمل من جانب الهواء.

1. INTRODUCTION

The compact Heat exchanger is a standard device to transfer heat between two fluids. It does utilize in vast industrial fields like power plants, automotive radiators, refrigeration, air conditioning, etc. [1]. Heat removal from the automotive engine is a substantial matter in achieving the best performance of the system[2]. Engineers and researchers must pay attention to improving heat transfer rates in automobiles, air conditioning systems, and freezing due to the increased scientific and technological growth throughout the world. Many studies have been conducted in this area to enhance the cooling system, such as modifying the design of the fins or tubes to increase the heating surface or enhancing the thermal characteristics of the working fluid.

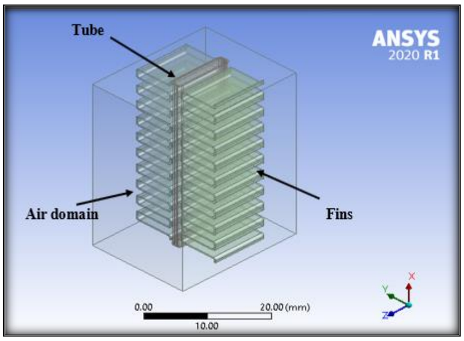
Priyadarsini et. al.[3] present a numerical analysis to enhance the radiator design and air-cooled system that included power plants, refrigerators, and internal combustion engines. The study found that circular fan sections had higher temperature drops and lower pressure drops, which could help to reduce the size of the cooling system. Abhilash et. al.[4]conducted a CFD simulation on an automotive radiator that would have been fixed and helical to determine the optimal model for transferring heat. Each model has been tested with aluminum dioxide, air, and water at 1.5 and 2.8 mass flow rates. The helical tube with an Al2O3/water mixture produced a greater reduction in outlet temperature and heat dissipation. Bhattarai et. al.[5] introduced a CFD simulation comparison of the behavior of a rectangular fin heat exchanger with a louver fin heat exchanger. The louver fin exhibited an increase in the rate the heat transfer. The radiator's weight was lowered by 6.27% with the use of louver fin.Zeeshan et. al. [6] involved compression between compact heat exchangers with flat and oval tube organized in line and staggered, respectively. An experimental result for a circular tube compact heat exchanger validates the predicted result. The study's findings suggested that the staggered oval tube proved to have a better increase in the air convective heat transfer coefficient (hair) under laminar flow, as well as discovered an increase in pressure drop with (hair).

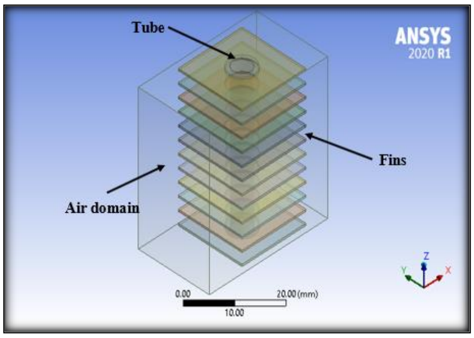
Borrajo-Peláez et. al.[7] intruduced a 3D numerical simulation of a heat exchanger using flat tubes and plain fins. To accurately predict the conduction heat transfer across tubes, fins, and water flow inside tubes while comparing the actions of the air and water sides, a new model was introduced. The effects of Reynolds number, fin (thickness, pitch, and length), and tube diameter were shown in the study of two models, air side and air water side. The results from two models were identical. The findings indicated that Nussult number raised with Reynold number, fin pitch and thickness, and lowered with fin length. An obvious outcome of increasing tube diameter is an improvement in Nusselt numbe.Myhren & Holmberg [8]Computational fluid dynamics was employed in this investigation (CFD). The operation entails altering the ventilation radiator's division, which combines heat emission and air supply to improve output heat transfer by lowering water temperature to maximize heat capacity and also save energy for heating system production and distribution using a heat pump .Oliet et. al.[9]there were two parts to the comparison CFD and (effectiveness -NTU) approach on car radiator, analysing the effects of the operating conditions (mass flow rate temperature difference, and fluid type) comes first, the second involved a geometrical analysis with parameters like (fin pitch and Louver angle). The result demonstrates that the heating rate and p were high whenever the louvre angle increased. The absence of design-changing studies to enhance performance while concurrently utilizing cooling fluids and nanomaterials was noted from the Previous studies .

In the current study, a cross-flow compact heat exchanger was created to match the bassline heat exchanger that is used in the automobile engine cooling system. It included changing the radiator's tube geometric features from a flat aluminum tube to a circular copper tube and also changing the louver fins to plain fins. The optimal diameter that provided the same tube cross-section area for the two models was also decided.

2. THE MATHMETICAL MODEL AND BOUNDARY CONDTIONS

The CFD model consists of two models, the baseline model and the designed model, which are used to analyze the thermal performance of small cross-flow heat exchangers. The dimensions of the two heat exchangers are shown in Table (1). Both geometries were produced using ANSYS FLUENT (2020 R1) and are made up of three components: a tube, water inside the tube, and an air domain. The geometry was then gathered into one section. The geometry of the two models is shown in Figure 1. Laminar flow occurs in the hot fluid at (75,85,95°C) at a mass flow rate of (10,12,18,24)flow rate .Table (2) illustrates the thermophysical properties of the hot mixture [10]. Air is the coolant fluid at 5.5 m/s and 35 °C. The thermal tube conditions are set as coupled, the outlet boundary conditions for both fluids are stationary motion for the wall and no-slip shear conditions. For the governing equation's solution convergence, the residual for energy equation, it is 10^{-6}. The highest number of iterations was 1000, with average to be between 400 and 800. The simulation process for the created model is shown in Figure 2 at (10) L/h hot mixture mass flow rate and (1.5,2.5,4.5,5.5) m/s cold fluid velocity.





(a) (b)

Figure 1. Geometry of Two Models. (a) Baseline model. (b) Designed model.

Table 1 The Dimension of The Heat Exchangers.

Heat exchanger type	Tube shape	Tube material	Fin shape	Tube length (m)	Inner diameter (m)	Outer diameter (m)	Tube cross section aera (m2)
Baseline model	elliptical	Aluminum	Louver	0.34	0.0103	10.7	1.74×10^{-5}
Design model	Circular	copper	Plain	0.34	0.0047	6.3	1.74×10^{-5}

Table 2 The Thermophysical Properties of Hot Mixture.

Density (kg/m^3)	Thermal conductivity (w/m. °C)	Specific heat (J/kg.k)	Dynamic viscosity (kg/m.s)
1042.04	0.392	3512	0.00107

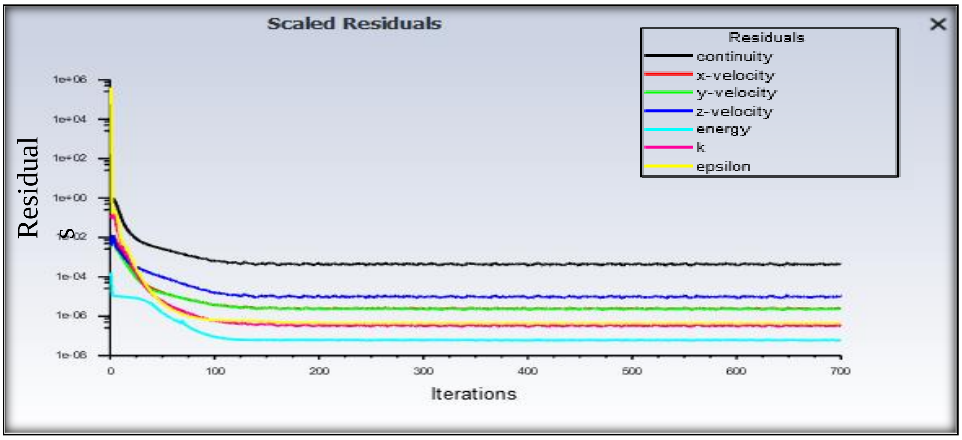


Figure 2 The Simulation Prosses.

3. GOVRRNING EQUATIONS

Assumption is applied in simulation which it is submit solve the governing equation and complete the simulation process. These assumptions include the flow under goes steady state conditions, The Physical properties constant with temperature, no heat loses, no heat generation, no phase change, The radiation effects are neglected and no mixing processes [11].

- **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

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

(2)

y-axis

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

z-axis

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

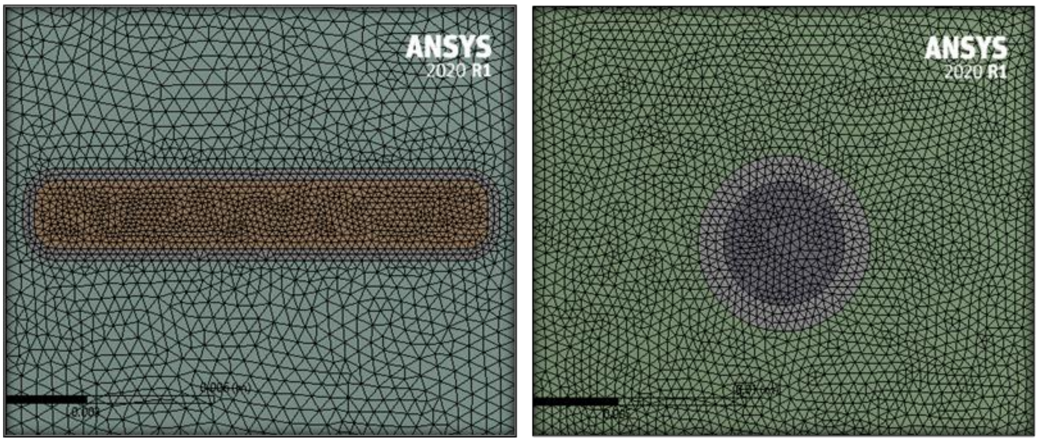
- **Conservation of Energy**

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

4. RESULTS AND DISCUSSION

4.1. Mesh generation and grid independency

Mesh generation is a significant simulation step because it imitates the geometry into a small cell (control volume) to improve the program's ability to solve partial differential equations. Mesh is divided into two types structured and unstructured.In this study, tetrahedral mesh was taken into account because it offers a more efficient solution for complicated geometry. The computational mesh for the investigation was displayed in Figure 3. Tetrahedral mesh quality was examined using the Skewness method; when the Skewness fine adjustment 1, the mesh's quality was acceptable reliability [12]. Overall number of nodes, elements, the skewness for this study as described in Table 3. For this study, the inlet boundary conditions have been obtained considering six different numbers of elements as showen in figure (4).



(a) (b)

Figure 3 Componential Mesh for The Two Models.(a) Baseline model.(b) Designed model

Table 3 Number of Nodes, Element and Skewness.

Type of H.E	No. of nodes	No. ofelements	Skewness
Baselinemodel	**823335**	**4804088**	**0.98797**
Designedmodel	**387531**	**2248512**	**0.84771**

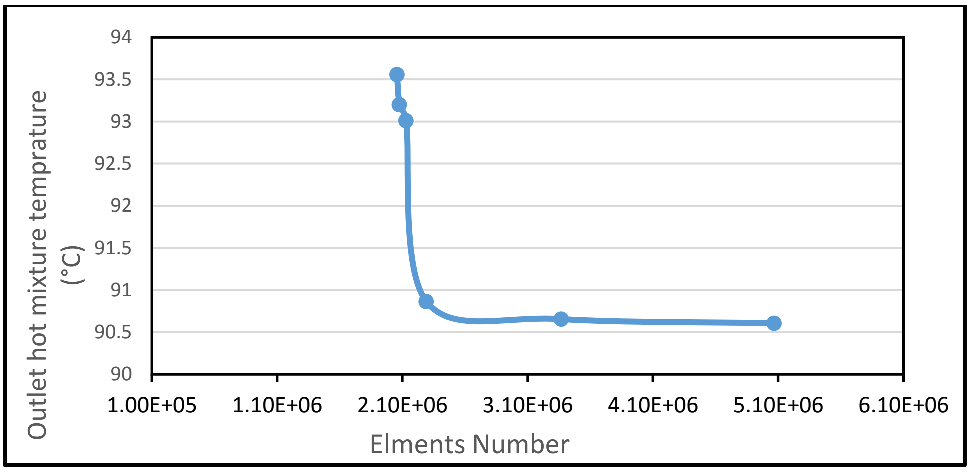


Figure 4 Grid independence test between number of elements and water outlet temperature in (°C).

4.2. Heat Transfer Rate

The compression of the heat transfer rate between the specified model with a 4.7 mm diameter and water mass flow rate is shown in Figure (5). Due to the circular shape of the tube, which expands the surface area and the better thermal conductivity of copper than aluminium, the designed model exhibited a significant advancement in heat transfer rate under the same boundary condition and inlet temperature for the two models (10 L/h hot mixture mass flow rate and 75 °C inlet temperature. The rate of heat transfer increased by a maximum of (18.6)%.

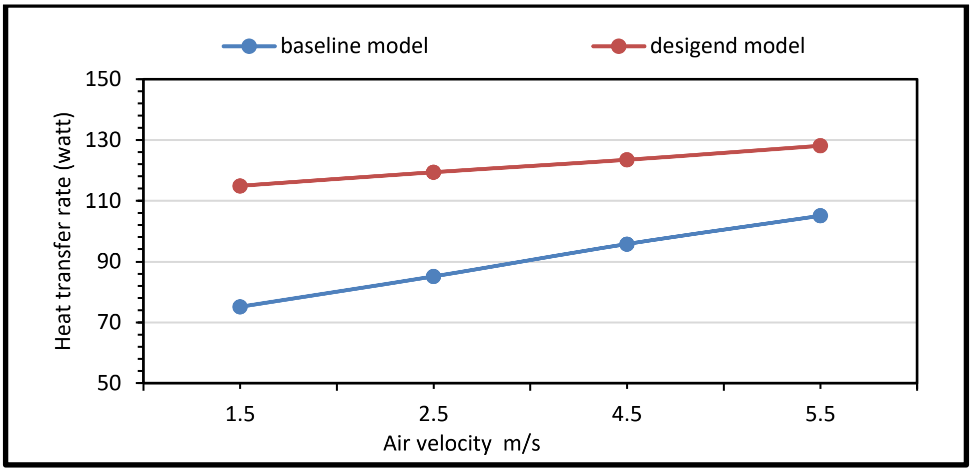


Figure 5 Heat transfer rate for baseline and designed model at 10L/h Hot mixture mass flowr ate and 75 °C inlet temperature.

4.2. Inner Side Overall Heat Transfer Coefficient (Ui)

Figure (6) shows how the baseline and designed models' overall heat transfer coefficients (Ui) on the water side are affected by air velocity and water inlet temperature. The result for the two models demonstrates that (Ui) rises with air velocity. In the designed model, plain fins allowed to unmix the flow, direct the flow, and penetrate the thermal boundary layer that forms on the walls. The designed model has enhanced the (Ui) by (13.84)% over bassline model.

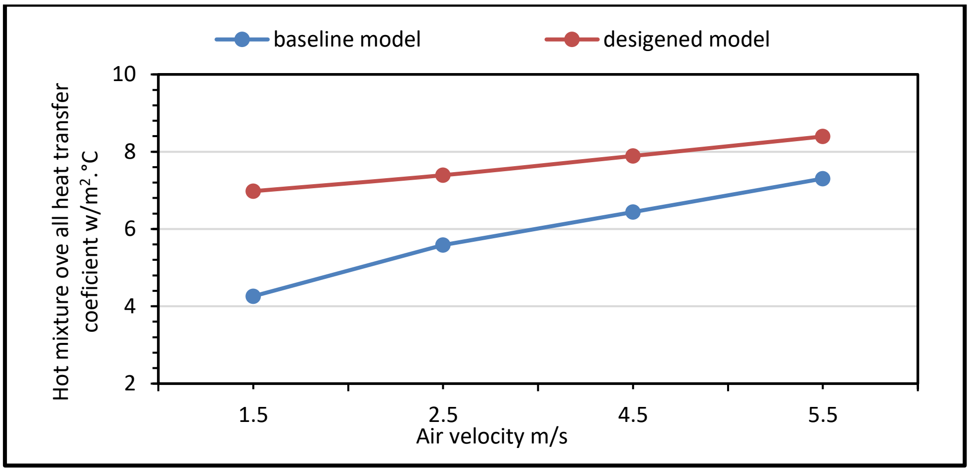


Figure 6 Inner side overall hat transfer corficient for baseline and designed model at 10L/h hot mixture mass flow rate and 75 °C inlet temperature.

4.4. Air side heat transfer coefficient (ho)

Figure (7) explains the air-side heat transfer coefficient for hot mixture mass flow rates of (10)L/h and (75) °C with 5.5 m/s air velocity for the bassline and designed model. It can be seen from this figure that the (ho) fore designed model is higher than the bassline model with a maximum enhancement ratio of (29.88) %.

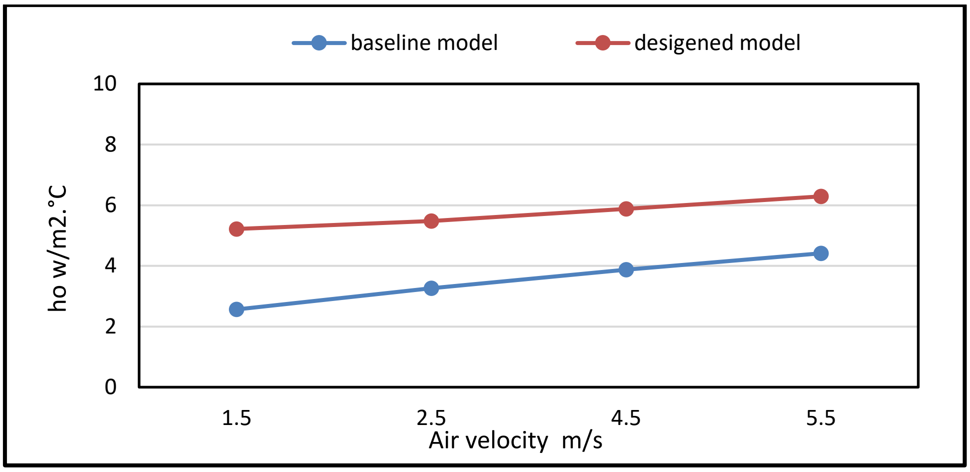
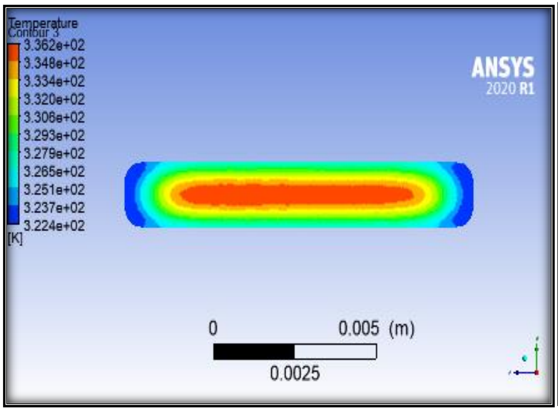
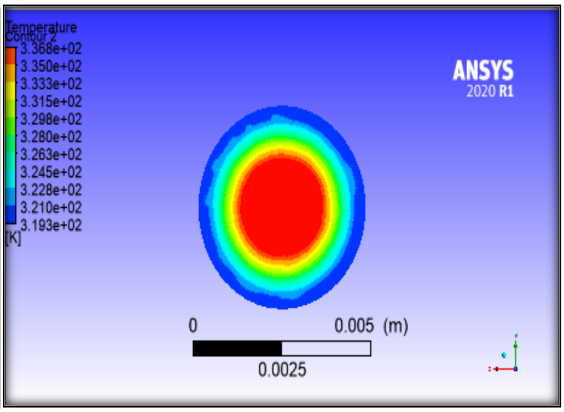


Figure 7 Air side heat transfer coefficient for baseline and designed model at 10L/h hot mixture mass flow rate and 75 °C inlet temperature.

4.5. Tamprature Countour

The hot mixture outlet temperature counter for the baseline and designed model for hot mixture mass flow rates of (10)L/h and (75) °C with 5.5 m/s air velocity of the current study is shown in Figure (8). The colours red and blue represent both high and low temperatures, respectively. Fins absorb heat transferred from the tube's wall through conduction and convection along the tube's length. The designed model's circular tube created channels for controlling water movement inside the tube, the baseline model's temperature distribution was higher near the inner surface of the tube. The temperature drops and heat transfer rate in the designed model appeared to be greater than in the baseline Figures (9 and 10) explain the temperature gradient on the tube for both model from the temperature distribution that the tube surface designed model hotter than the bassline this means there is good exchange between the hot mixture and tube surface.





(a) (b)

Figure 8 Temperature distribution for hot fluid bassline model for (a) Baseline model. (b)Designed model.

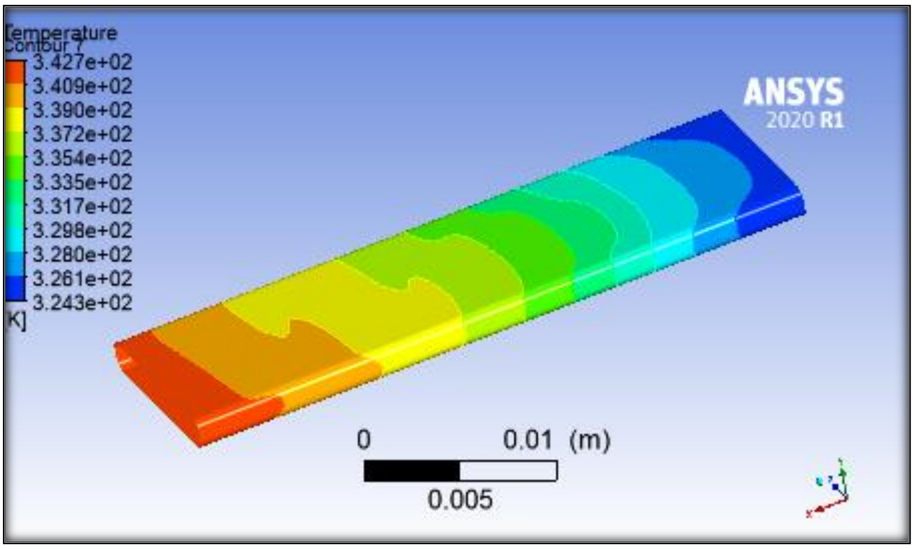


Figure 9 Temperature distribution of the tube for bassline model.

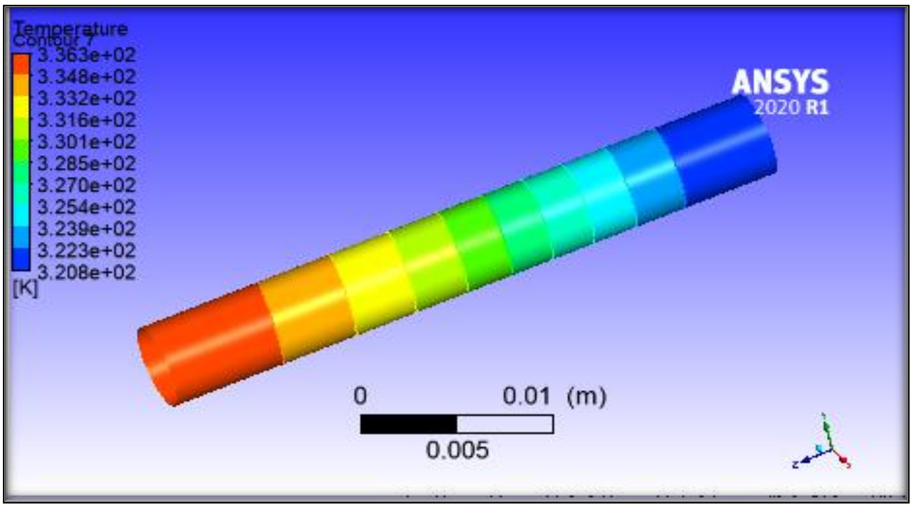


Figure 10 Temperature distribution of the tube for desigened model.

6. CONCLUSIONS

1. A CFD simulation was applied in the study to evaluate the thermal behaviour of a baseline compact heat exchanger with louver fins on a flat tube to a newly created model with a circular tube and plain fins based on the same dimensions. The flowing findings have been reached. The optimum inner tube diameter that gives the same tube cross section area of the bassline model was 4.7 mm.
2. The circular tube gave an improvement in thermal performance compared to the flat tube.
3. This numerical result found that changing the material of the tube from Aluminum to Copper increased the temperature drop because of the high thermal conductivity of the copper than Aluminum.
4. The designed model shows an enhancement in heat transfer rate over the baseline model with a maximum percentage of increment (18.6) %.
5. The overall heat transfer coefficient in the designed model was higher than the bassline model by (13.8)%. as the temperature difference increased.

6. As result of temperature drop in hot mixture air side heat transfer coefficient increased. The maximum increasing is (29.8) %.

REFERENCES

1. Karimi A, & Afrad M, (2018). Numerical study on thermal performance of an air- cooled heat exchanger: Effect of hybrid nano fluid, pipe arrangement and cross section. *Energy convection and management*, vol.164, pp. 615–628. https://doi.org/10.1016/j.enconman.2018.03.038
2. Tijani A. S, & Sudirman A. S. bin, (2018). Thermos-physical properties and heat transfer characteristics of water/anti-freezing and Al2O3/CuO based nanofluid as a coolant for car radiator. *International Journal of Heat and Mass Transfer*,vol.(118), pp. 48–57. https://doi.org/10.1016/j.ijheatmasstransfer.2017.10.083
3. Priyadarsini, C. I., Reddy, T. R., & Devi, P. A. (2022). Design and Performance Analysis of Automotive Radiator using Computational Fluid Dynamics. *In International Journal of Mechanical Engineering* ,Vol. 7, Issue 3.
4. Abhilash, P., Raghupathi, U., & Kumar, P. (2020). Design and testing of radiator with fixed channel and helical pipe using nanofluids. *Materials Today*: Proceedings, 39, 615–620. https://doi.org/10.1016/j.matpr.2020.09.002
5. Bhattarai S, Vichare P, Dahal K, Ghimire SK, & Raju KM, (2017). Computational Fluid Dynamics based Performance Evaluation of Louvered Fin Radiator. *In Proceedings of the 11th International Conference on Software, Knowledge, Information Management & Applications*. Sri Lanka Institute of Information Technology. Sri Lanka.
6. Zeeshan M., Nath S., & Banja, D. (2017). Numerical study to predict optimal configuration of fin and tube compact heat exchanger with various tube shapes and spatial arrangements. *Energy Conversion and Management*, 148, 737–752. https://doi.org/10.1016/j.enconman.2017.06.011
7. Borrajo Peláez R., Ortega Casanova J., & Cejudo-López J. M., (2010). A three-dimensional numerical study and comparison between the air side model and the air/water side model of a plain fin-and-tube heat exchanger.*Applied Thermal Engineering*, Vol.30(13), pp.1608–1615. https://doi.org/10.1016/j.applthermaleng.2010.03.018.
8. Myhren J. A., Holmberg, S. (2011). Improving the thermal performance of ventilation radiators - The role of internal convection fins. *International Journal of Thermal Sciences*, Vol.50(2), pp. 115–123. https://doi.org/10.1016/j.ijthermalsci.2010.10.011
9. Oliet C., Oliva A., Castro J., and Pérez-Segarra C. D., (2007). Parametric studies on automotive radiators. *Applied Thermal Engineering*, Vol 27(11–12), pp.2033–2043. https://doi.org/10.1016/j.applthermaleng.2006.12.006
10. American Society of Heating, R. and A.-C. Engineers. (2009). ASHRAE handbook: fundamentals. American Society of Heating, Refrigeration and Air-Conditioning Engineers.
11. Taher F. A. and Kadhim Z. K., (2019). Experimental and Numerical Investigation of Different Finned Tube Heat Exchanger M.Sc. thesis, The University of Wasit. Eng. collage, Mech. Dep, Iraq.
12. Mardan S. A. and Kadhim Z. K., (2020). Numerical and Experimental Study of Cross-Flow External Finned Tubes Heat Exchanger. M.Sc. thesis, The University of Wasit. Eng. collage, Mech. Dep, Iraq.