International Information and Engineering Technology Association
Advancing the World of Information and Engineering

International Journal of Heat and Technology
Vol. 42, No. 4, August, 2024, pp. 1219-1230
Journal homepage: http://iieta.org/journals/ijht

Exploring the Impacts of System Geometry on Heat Transfer Efficiency in Coil-and-Tube Heat Exchangers

Jaafar Saleem[1], Khaled M.N. Chahrour[1], Laith Jaafer Habeeb[2*]

[1] Mechanical Engineering Department, Faculty of Engineering, Karabuk University, Karabuk 78050, Turkey
[2] Training and Workshop Center, University of Technology- Iraq, Baghdad 10066, Iraq

Corresponding Author Email: Laith.J.Habeeb@uotechnology.edu.iq

https://doi.org/10.18280/ijht.420412

Received: 7 September 2023
Revised: 19 June 2024
Accepted: 17 July 2024
Available online: 31 August 2024

Keywords:
coil-and-tube heat exchanger, ANSYS simulation, coefficient of performance, temperature and velocity contour

ABSTRACT

The purpose of this research is to investigate in depth the effect of system geometry on the performance of coil-and-tube heat exchangers. This work investigates the use of direct evaporative cooling (DEC) to improve the efficiency of air conditioning systems in extremely hot weather. DEC systems must be sized and constructed properly for the particular building and climate in which they are going to be installed. The ultimate goal is to reduce energy usage while producing affordable, green cooling options for regions with exceptionally hot temperatures. For simulation, we used computational fluid dynamics with ANSYS software. We purposefully varied the quantity of nozzles and the number of turns in the cooling tubes to see how they affected the effectiveness of the system. The temperature and velocity plot findings revealed a substantial difference, demonstrating that these engineering elements had an impact on the heat exchange process. According to the results, setups with 35 nozzles and 13 nozzle turns had a maximum coefficient of performance of 4.537 at an entry velocity of 2 m/s, showing that this configuration is ideal in the investigated conditions. The findings emphasize the need for strategically modifying some engineering parameters to increase heat exchanger performance and offer exciting prospects for the design and optimization of such systems. More research is needed to better understand these complicated interactions, including the investigation of different fluids, channel diameters, pressure conditions, and performance in the transient state.

1. INTRODUCTION

Heat exchangers are involved in the transfer of energy in different functions in industries and commerce since they act as critical channels. The design and optimization of these exchangers are important because they directly affect the energy aspect, the reliability in operation, and the performance of the system. Therefore, the investigation of the factors affecting the efficiency of heat exchangers goes on. The main aim of this research is to determine the effect of system engineering of coil-and-tube heat exchangers. The number of turns in the cooling tubes and the number of nozzles used in the system were altered systematically in a computational fluid dynamics simulation done by the ANSYS software. Temperature and velocity distribution effects were then looked at. The findings make out the interdependency of these geometric characteristics with the thermodynamics of the system with the objective of enhancing the performance of the exchanger. Thus, this work provides the basis for the further improvement of the understanding of the design of coil-and-tube heat exchangers and the further increase of efficiency of the heat exchange system. As a result of these problems, scholars have been searching for less energy-intensive and more eco-friendly solutions to the conventional air conditioning systems. One of these options is the direct evaporative cooling (DEC) that has received some attention because of being energy and cost efficient. The focus of this literature review will be to describe the employment of DEC in enhancing AC systems in exceedingly high temperatures and show the advantages and disadvantages of DEC, mathematical modeling approaches, and optimization methods for different applications. Some proposal of how DEC systems should be designed and how they should function will be given and then DEC systems will be compared with the conventional air conditioning systems.

2. LITERATURE REVIEW

EC solutions involve the use of sustainable and renewable energy devices and can substitute HV AC in hot and dry climates. The following article gives information about the recent developments, issues, and prospects of EC systems.

The research process starts with the work of Camargo et al. [1], who described an integrated system based on EC and vapor pressure and conducted sensitivity analyses of the system variables using a system model. In another study, Eidan et al. [2] studied the ability of DEC systems to conserve energy and according to them, the systems substantially consumed lesser power in heating ventilation and air conditioning especially in areas characterized by aridity. Other research focuses on improving the performance of different EC systems. Zhou et al. [3] conducted an experiment to improve the performance of a DEC and found that the air velocity in wet ducts can affect the cooling efficiency.

New applications and specific scenarios have been studied in several papers. Mousavi et al. [4] mentioned the application of an EV system in a sustainable farming chamber in the United Arab Emirates. As for the outcomes of their experiments, they proved that under certain conditions the temperature was decreased to 7-16℃ with the help of the system. Many simulated theoretical studies were greatly enhanced by the knowledge concerning the enhancement of the performance of evaporative coolers. In the study of Moshari and Heidarinejad [5], various types of ECs were modeled numerically, and it was determined that the counter-flow is the highest in terms of humidity ratio, and the lowest incoming air temperature.

Moreover, literature also provides understanding about the impact of different variables on the effectiveness of EC systems. According to Adam et al. [6], it was established that high WOF improves the cooling capability and the wet point effectiveness. In a study by Fan et al. [7], it was found that the mechanical vibration and magnetic fields could pose quite a great impact on the EC process in space stations. Some of the special designs aimed at enhancing the performance of indirect EC (IEC) systems were also described in the review. Jradi and Riffat [8] developed psychrometric core, new strategy that has prospect for providing thermal comfort in buildings.

Cui et al. [9] described a special type of evaporative air conditioner that works with dew point that chills the air to a dew point temperature, making it an additional effective option for cooling. Zhou et al. [10] proposed a new system named collaborative cross-current DEC, which can enhance the performance with both thermoelectric and IEC. Khafaji et al. [11] have numerically investigated planar forced combustion in an evaporation cell and found that maximum of this value depends on the change of the features of the cell. Riangvilaikul and Kumar [12] demonstrated the usefulness of an evaporative dewpoint cooling system in the sensory cooling of vented air particularly in hot and dry climates. Sheng and Nnanna [13] investigated the influence of different parameters of the system on DEC and found that the cooling capacity is a strong function of the dry front air temperature and an inverse function of the front air velocity and the inlet water temperature. Majdi et al. [14] devoted to the analysis of the use of plate heat exchangers for heat exchange with the main advantage of high thermal efficiency. They analyze the effect of the increase in the plate number on the transfer area and heat energy transfer where the highest performance is obtained under 308.1 K. Banooni et al. [15] enhanced the mini-channels of plate-fin heat exchangers using numerical analysis and modeling for the corrugated fin. The analysis reveals that temperature distribution affects the velocity of water and air that in turn affects pressure difference and thermal effectiveness. Hameed Hasan et al. [16] investigated in a CFD analysis for the flow field, thickness, wavelength, and boundary layer, the equations of numerical analysis were applied. Variations in temperature with reference to the depth of heat exchanger were due to convection and conduction heat transfer. Saleem et al. [17] focused on energy saving and eco-friendly cooling techniques through simulation and analysis with the help of CFD tool. It also has adjustable injectors and coil turns for the external air cooling. The best configuration of the injectors was 35 while that of the coil turns was 13, resulting in a COP of 4.537, which proves that geometry indeed has a great influence on performance.

Studies indicate that EC solutions are a more energy-efficient and environmentally friendly alternative to typical HVAC systems, particularly in dry settings. However, the system's design and operation must be meticulously optimized to realize its full potential. So far, significant progress has been made in this field. It is revealed from the studies provided in this paper that the EC technologies have the potential to offer alternative and sustainable and energy efficient solution to HVAC systems. In hot and dry circumstances, the application of Solid-Desiccant IEC in wet conditions significantly reduces the buildings' energy consumption, and DEC and IEC systems. The research also puts much emphasis on how the design and operational characteristics of the systems need to be enhanced to the highest levels of efficiency and performance. These factors include water flow rate, air velocity, and the temperature and humidity of the input air, and the design of the EC system.

The research evidence shows that the EC systems have a bright future in the construction of energy-efficient sustainable buildings. However, further research has to be conducted in order to determine all the benefits and improve their performance for various structures and climates. According to the research elucidated in the above section, the DEC approach is a feasible way of reducing the energy consumption of HVAC systems. The studies reported that front air velocity, dry bulb temperature and the water temperature that enters the cooling tower influenced the cooling rate of the tower and functional relations between the variables were determined. Stud have also been conducted on how the type of wetted media affects the efficiency of DEC and the application of pre and post cooling sections in the DEC system to increase the efficiency. Therefore, taking into account all the analyzed materials, DEC can replace the conventional HVAC systems that are characteristic of dry and desert climates.

3. METHODOLOGY

This paper uses a quantitative and simulation approach to assess the impact of heat exchangers on the energy effectiveness and performance of refrigeration systems. The methodology involves these concepts in thermodynamics, fluid mechanics, and heat transfer, with the help of the EES and ANSYS simulation and analysis software packages.

3.1 Software tools and general equations

In this thesis, the Engineering Equation Solver (EES) software is used intensively as well as the ANSYS software package. EES is widely used by engineers to solve complex engineering problems related to heat transfer, fluid mechanics, thermodynamics and other course. This tool is equipped with user-friendly interface to work with as well as with a lot of equations and thermal characteristics included into its base [4].

We perform detailed CFD analysis with the help of the ANSYS software package which helps in generation of system geometry, grid mesh generation for simulations and execution of simulations. Thus, we are using the k-ϵ turbulence model in ANSYS since it is effective for most turbulent flows.

Besides the software tools, the basic principles of mass conservation, the first law of thermodynamics and entropy are applied to analyzed systems.

The equation of mass conservation for the system is provided as follows:

$$\sum \dot{m}_{in} = \sum \dot{m}_{out} \tag{1}$$

where,

$\sum \dot{m}_{in}$: the entire mass entering in a certain amount of time.

$\sum \dot{m}_{out}$: the entire mass exiting in a certain amount of time.

The first law of thermodynamics for the system serves as the foundation for the energy balance for each component:

$$\dot{Q} + \dot{W} = \sum \dot{m}_{out} h_{out} - \sum \dot{m}_{in} h_{\text{in}} \tag{2}$$

where,

$\dot{Q}$: the heat transfer in a given amount of time.

$\dot{W}$: work completed in a unit of time by the control volume.

h_{in}: certain enthalpy for each mass that enters the system.

h_{out}: certain enthalpy for each mass that leaves the system.

Entropy is created via irreversibility. Thus, it is not conserved in open or closed systems, in contrast to mass and energy. The entropy balance in open systems is:

$$\dot{E} = \dot{m}\psi \tag{3}$$

The continuity, momentum, and energy equations are the governing equations that need to be solved.

Conservation of mass:

$$\nabla \cdot (V) = 0 \tag{4}$$

Equation of momentum:

$$\nabla \cdot (\rho \dot{V} \dot{V}) = -\nabla p + \nabla \cdot (\bar{\tau}) \tag{5}$$

Stress tenser $\bar{\tau}$ is given by:

$$\bar{\tau} = \mu \left[\left(\nabla \vec{V} + \nabla \vec{V}^T \right) - \frac{2}{3} \nabla \cdot \vec{V} I \right] \tag{6}$$

Equation of energy:

$$\nabla \cdot (\dot{V}(\rho E)) = \nabla (k \nabla T - \rho C \dot{V} T') \tag{7}$$

The predictive capabilities of the heat transfer processes are mainly based on the mass, energy, and entropy balance equations.

The law of conservation of mass which states that mass cannot be created or destroyed is depicted by the conservation of mass equation. In the context of a heat exchanger, it ensures that the mass flow rate of the fluids coming into and out of the heat exchanger are equal. Thereby, it aids in the understanding of the fluid dynamics and the aspects of the system such as heat transfer.

The energy balance consideration takes the system's energy conservation into account. It is useful in the determination of heat transfer from the fluid to the inner surface of the tubes under heat exchanger conditions. It incorporates convective heat transfer coefficient of the fluid and the tube wall, thermal conductivity of the wall, and any heat generation or removal within the system. The design of the heat exchanger requires that the total energy transfer and the temperature pattern should be well understood.

The change of entropy of the system is calculated from the entropy balance equation. This equation helps a better understanding of entropy change when heat is transferred. Entropy is used to calculate the amount of randomness of the system. Within the context of the heat exchanger, it offers information on the irreversible and loss processes with the heat transfer. Hence to maximize the heat exchanger efficiency, it becomes necessary to minimize the entropy formation.

These formulas provide the starting point in making mass transfer, energy transfer, and entropy production analysis of the coil-and-tube heat exchanger. The use of these conservation equations will help scholars to understand the heat transfer processes and make sound decisions with an aim of improving the efficiency of the system.

3.2 System design and solution parameters

Thus, our system's design focuses on the thermal effect of the heat exchange process in which the coil-and-tube heat exchanger contains a variable number of turns. The system design also includes a variable-number nozzle water spraying procedure and an air entry space. The structure of the system is presented in the Figure 1. A tetrahedron grid was used to generate the mesh because of its superior performance with complex geometries. A precise mesh is necessary for the simulation process to produce reliable results.

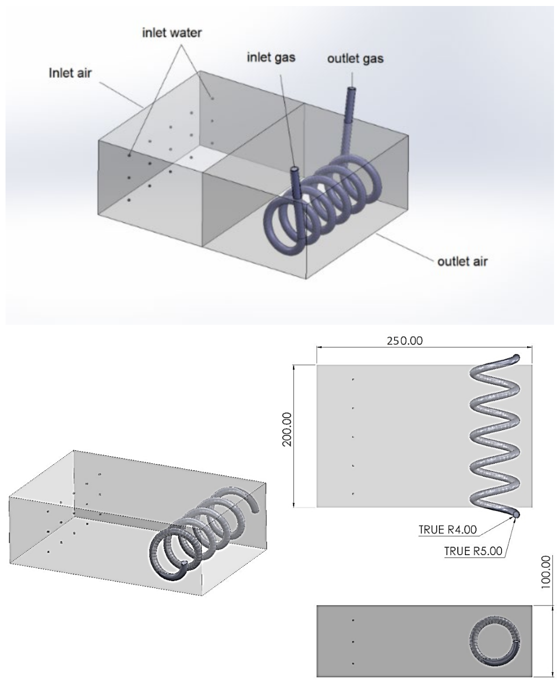


Figure 1. Geometry shape

Using the pressures and temperatures extracted from the EES program, which we also used as input for the CFD program in ANSYS, we determined the boundary conditions for our study. Initial conditions were established relative to the boundary conditions at the inner pipe's inlet.

The solution parameters, such as the type of precision solver and the number of iterations, were determined to produce precise results. Our solution's dependability was ensured by setting convergence criteria at an error residual level of 10^{-6}.

The complicated geometry employed a tetrahedron-unstructured grid. A phase is input in ANSYS to produce the 3D model mesh. Matrix and equation solutions require a precise mesh. Multiple stable meshes were tested. On the basis of 3,701,221 elements, the average outlet temperature was 53.903 m/s, confirming mesh independence as shown in Table 1.

This combination of software tools, governing equations, system design, and solution parameters has enabled a thorough investigation of heat exchangers in refrigeration systems, yielding insights into their performance and optimization.

Table 1. Mesh independency

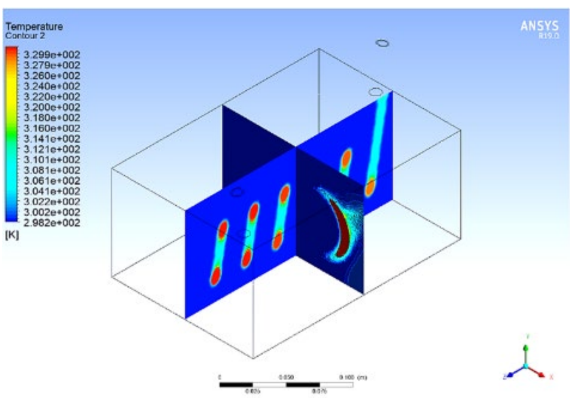
Cases	Elements	Nodes	Gas Temperature °C
1	3025365	623482	53.964
2	3412354	734210	53.909
3	3701221	877558	53.903

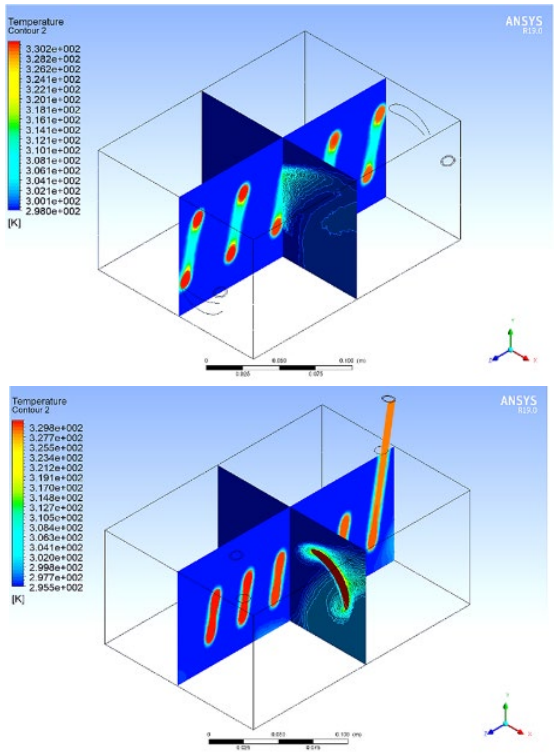
4. RESULTS AND DISCUSSION

4.1 Analysis of temperature contours

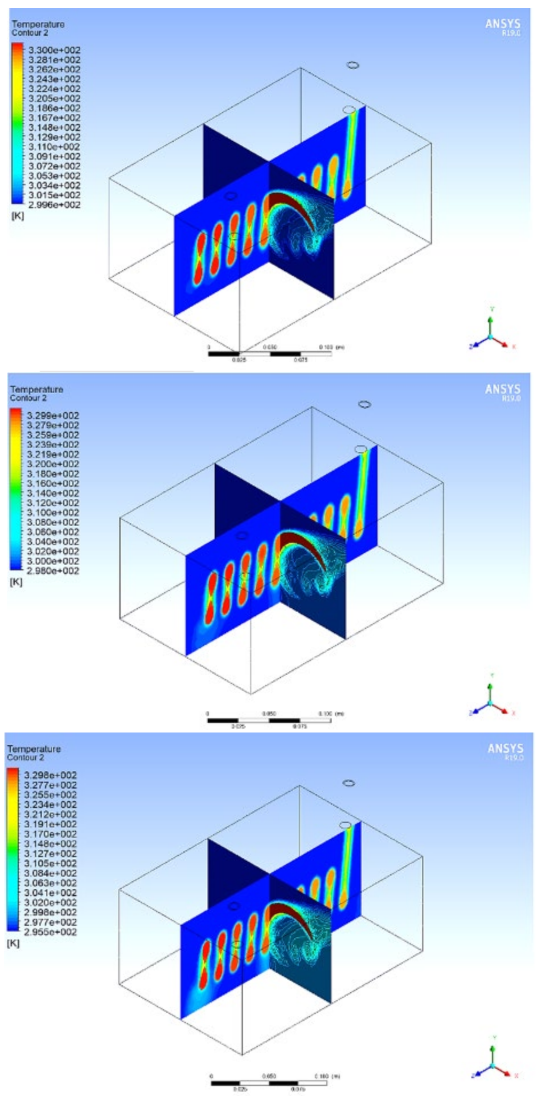
CFD analysis requires understanding how system architecture affects heat transfer performance. This study examined how changing the number of nozzles and cooling tube turns affects the performance of a coil-and-tube heat exchanger. A total of 15-35 nozzle counts and 5-13 cooling tube turns were tested. ANSYS used an unstructured tetrahedron grid in solving the geometries of the system. The following findings and discussion were based on this technology that recognised trends in the temperature and velocity contours which came from the changes.

Some temperature contour variations within a system are illustrated in the Figures 2 to 4 and these include different nozzle types, cooling tube turns as well as inlet velocities of 0. 5, 1, and 2 m/s winds are implied. It is found from the ANSYS simulation that, changes in number of turns and nozzles lead to changes in the temperature contour. The maximum heat distribution is observed in the case when the system comprises of 13 turns and 35 nozzles. From this observation, one can derive that with higher number of turns and nozzles, it is feasible to improve the heat exchange and temperature distribution, thus the efficiency of the system. There may be possibilities of increasing the heat exchange when changes have been made to the design.

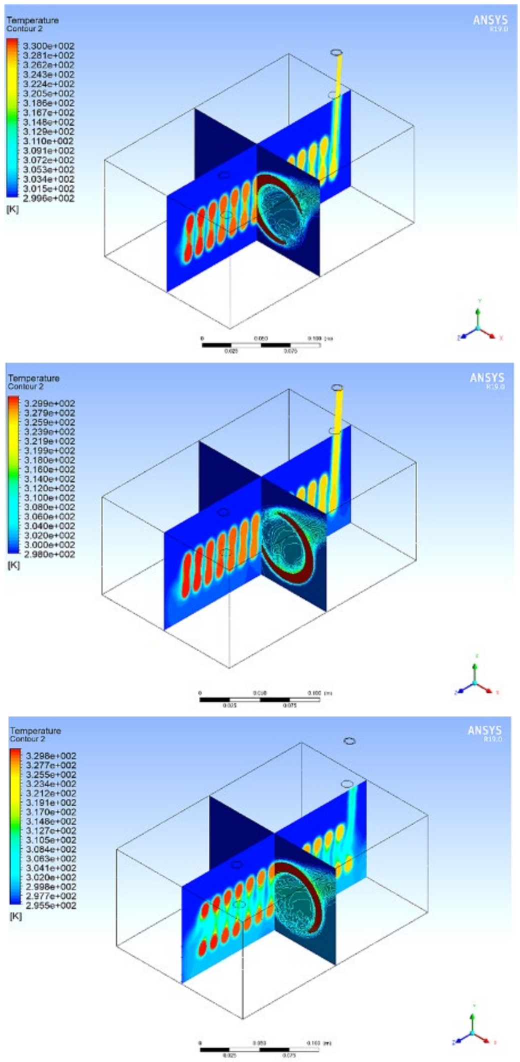




(a) 5 turns

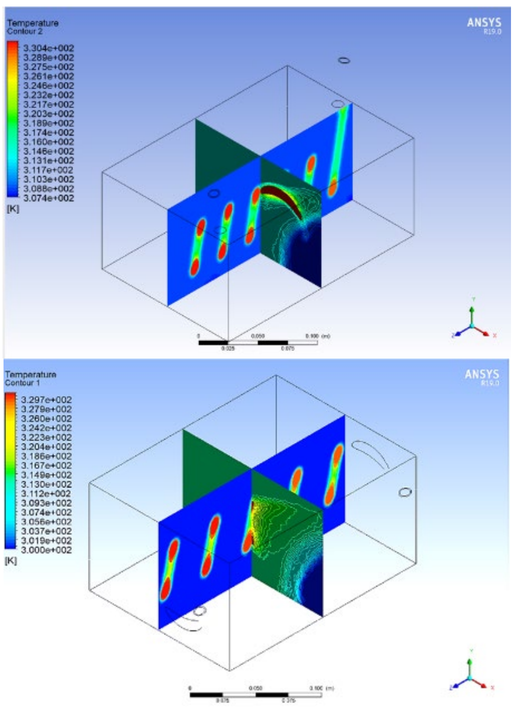


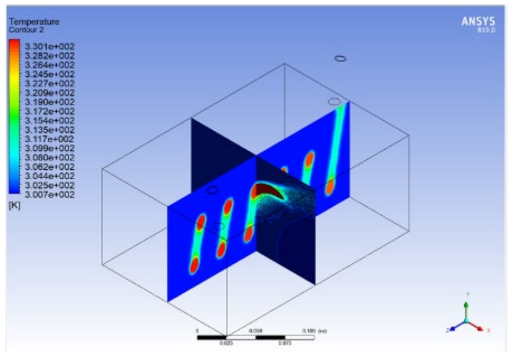
(b) 9 turns



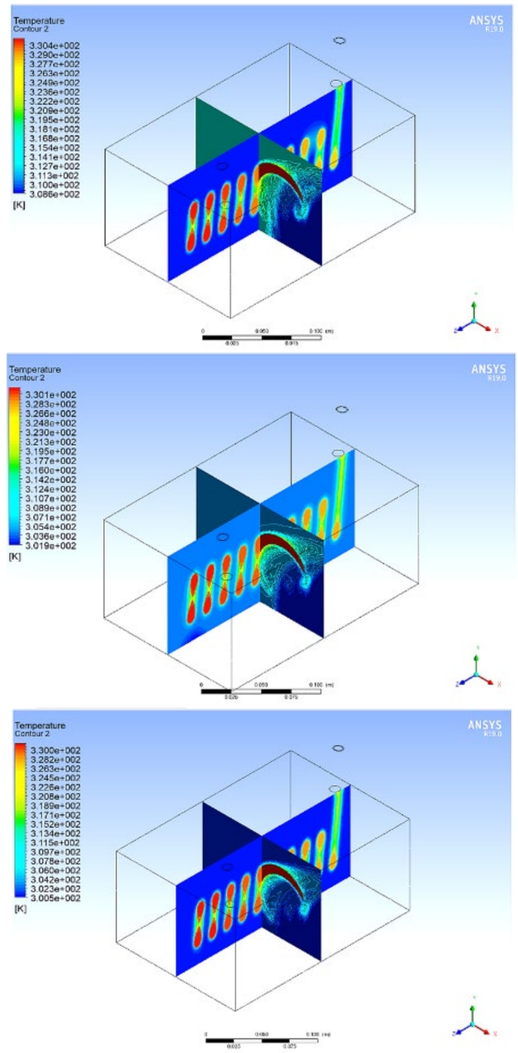
(c) 13 turns

Figure 2. Temperature profile analysis for a system at inlet velocity 0.5 m/s with 15, 24, and 35 nozzles with 5, 9, and 13 turns

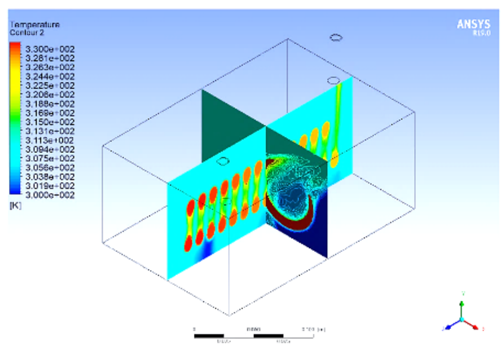


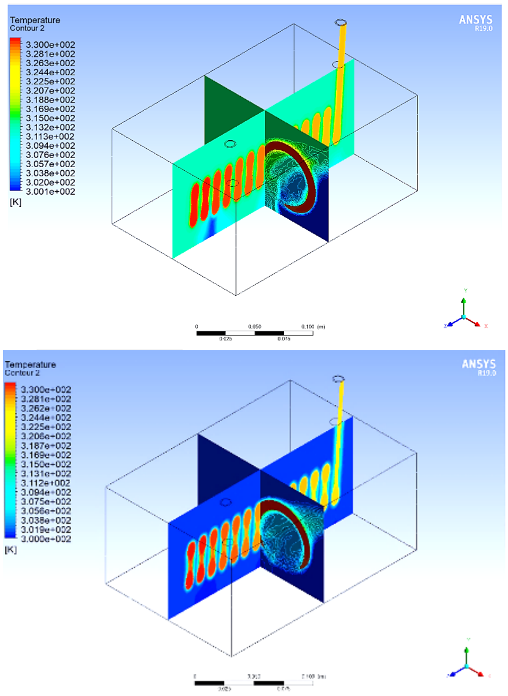


(a) 5 turns



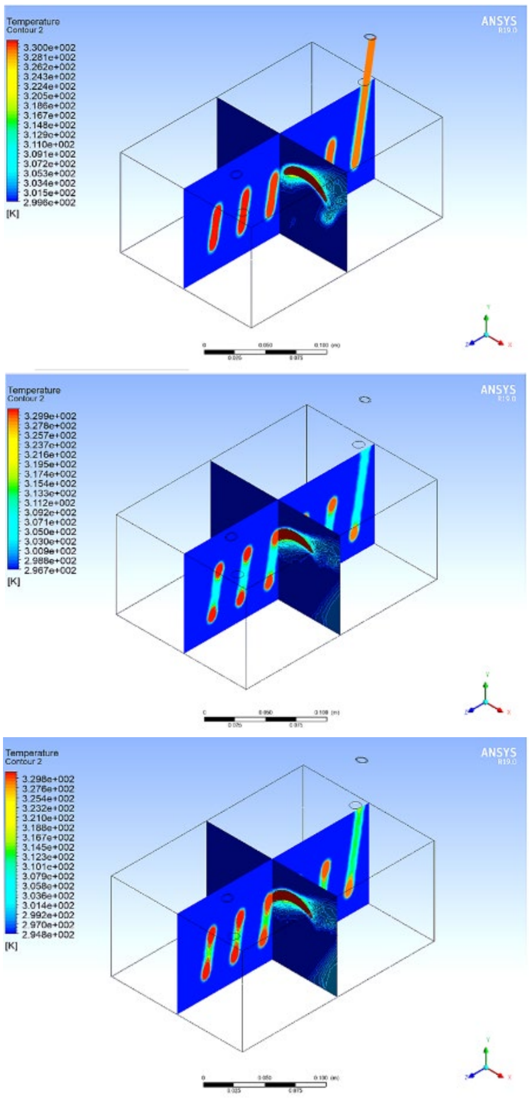
(b) 9 turns



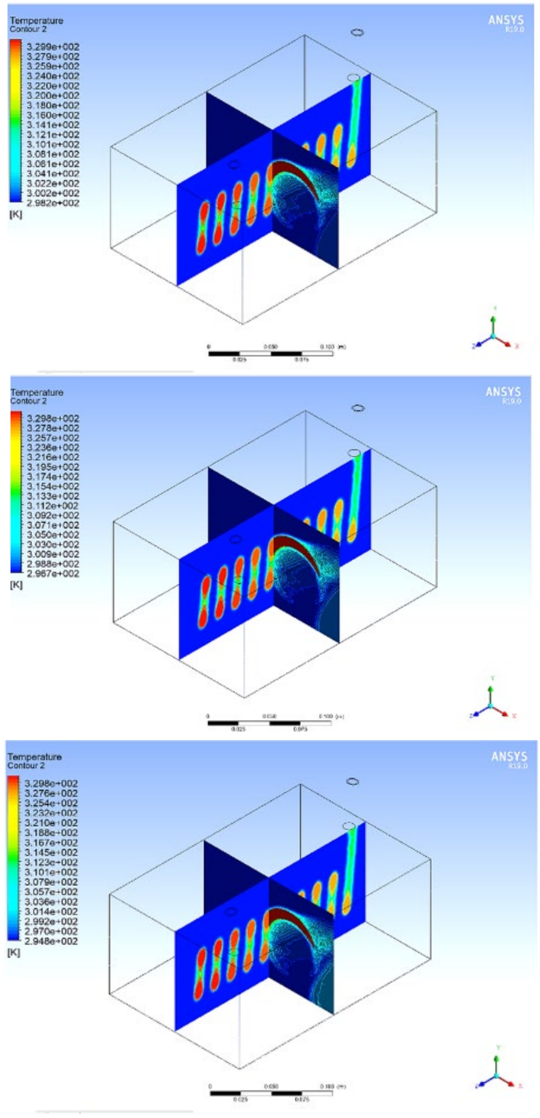


(c) 13 turns

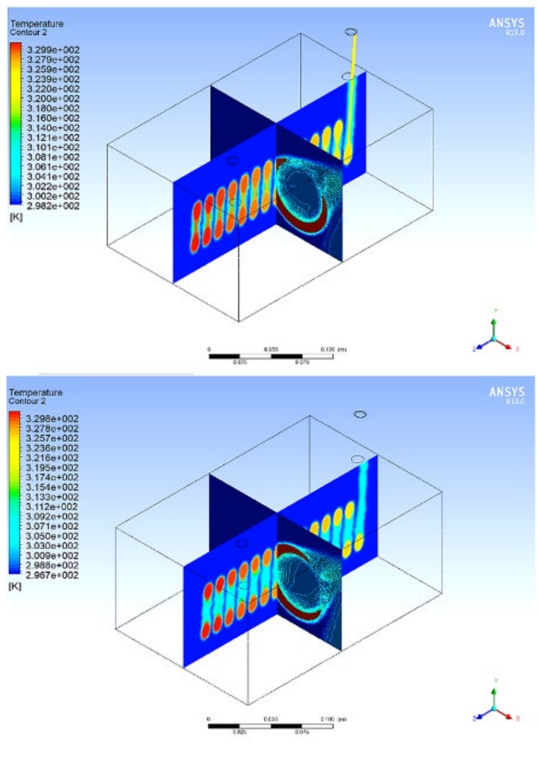
Figure 3. Temperature profile analysis for a system at inlet velocity 1 m/s with 15, 24, and 35 nozzles with 5, 9, and 13 turns

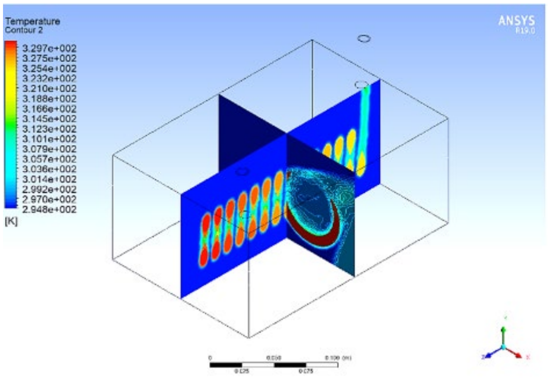


(a) 5 turns



(b) 9 turns





(c) 13 turns

Figure 4. Temperature profile analysis for a system at 2 m/s inlet velocity with 15, 24, and 35 nozzles and 5, 9, and 13 cooling tube turns

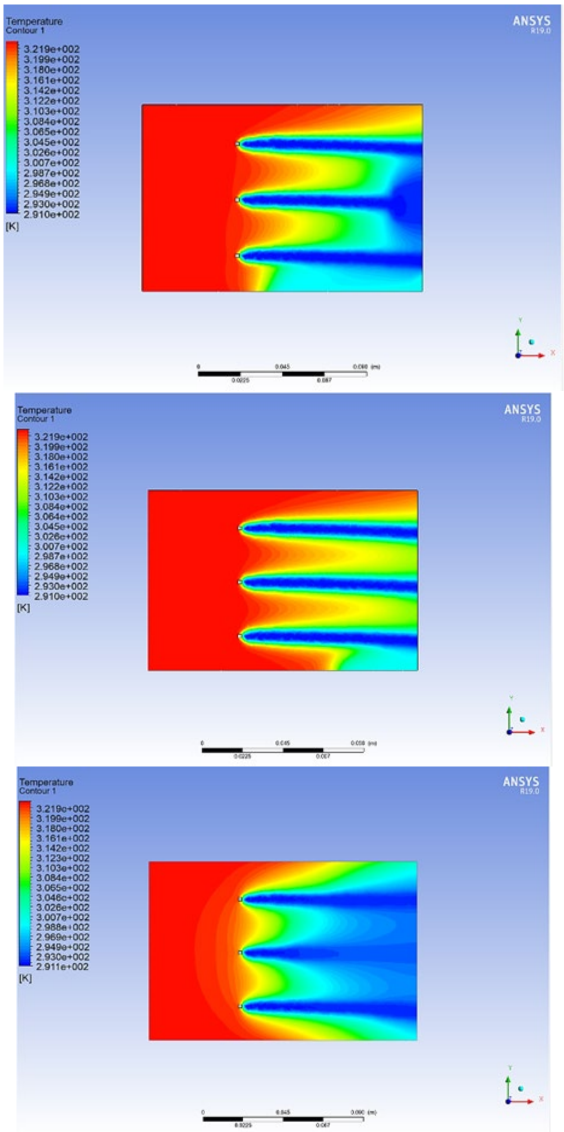
The above work show that the increased surface area is for the results that were observed to facilitate heat transfer and the multiple tubes, turns, and nozzles. More turns lengthen the channels through which fluids are exchanged and more nozzles enhance the spreading of the fluids. However, there are potential drawbacks which have to be taken into consideration, for example, one of them is a potential for higher pressure drop and the other one is the complexity of design. Further studies should focus on fine-tuning these parameters to achieve the best design at the expense of other factors if need be.

The total heat transfer surface area is increased by adding nozzles, which may improve heat transfer efficiency. There is less chance of hotspots or regions with insufficient heat exchange when more nozzles are present because the fluid distribution throughout the tubes becomes more consistent. Greater numbers of nozzles result in greater fluid velocities, which improve heat transmission, depending on the flow rate and design. Reducing the number of nozzles streamlines the design, which might facilitate production and maintenance. Lower heat transfer efficiency and uneven fluid distribution result from fewer nozzles, especially if the fluid does not sufficiently cover the whole heat exchange surface.

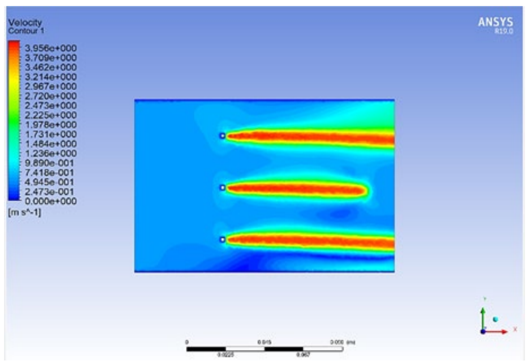
Whereby more tube turns increase the heat exchange path, longer contact with the fluid tube surface is achieved. This may enhance heat transfer since it takes a longer route. There is improved heat transfer within the tubes due to the creation of turbulence by the added twist. As per the flow path, pressure drop and pumping power requirements can be reduced by reducing the number of tube turns. However, less number of turns of tubes decreases the length of the heat exchange and decreases the time for which the fluid and the inner surface of the tube interact, thus may affect the efficiency of heat transfer.

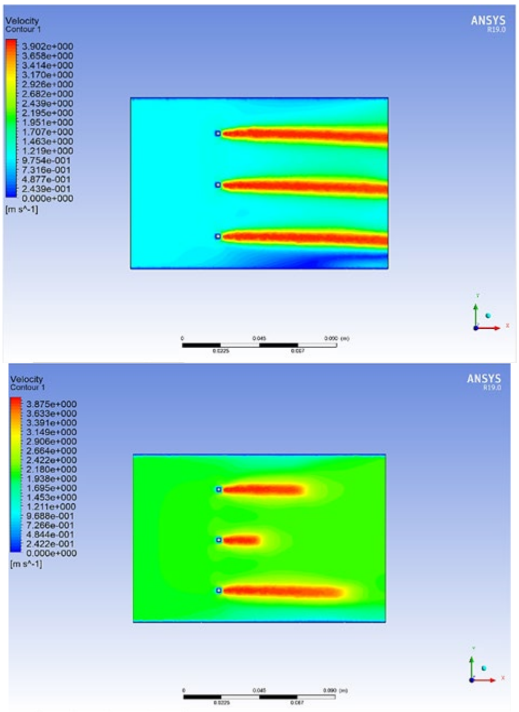
4.2 Effects of water spraying settings and air entrance geometry on thermal dynamics

Figures 5-7 show changes in temperature and velocity contour in relation to the changes in air velocity and nozzle number. The results proved that the temperature distribution inside the system is consistent and there is no extreme temperature variation in any part of the system. Reducing the velocity results into a more uniform spread of heat because of the longer time that the air takes within the system. This extended period allows adequate heat exchange with the cooling tubes. When the velocities are higher, the temperatures also show a gradient which indicates that the heat transfer is reduced due to the short duration of contact between the air and the cooling tubes. Further, it is necessary to point out that the changes in nozzle designs impact the distribution of temperature. The number of nozzles is 35, and by their means, the distribution of temperature is even more correct than with configurations of 15 or 24 nozzles. This finding indicates heightened efficiency in the heat exchange process, which is attributable to the augmented contact area between air and water.



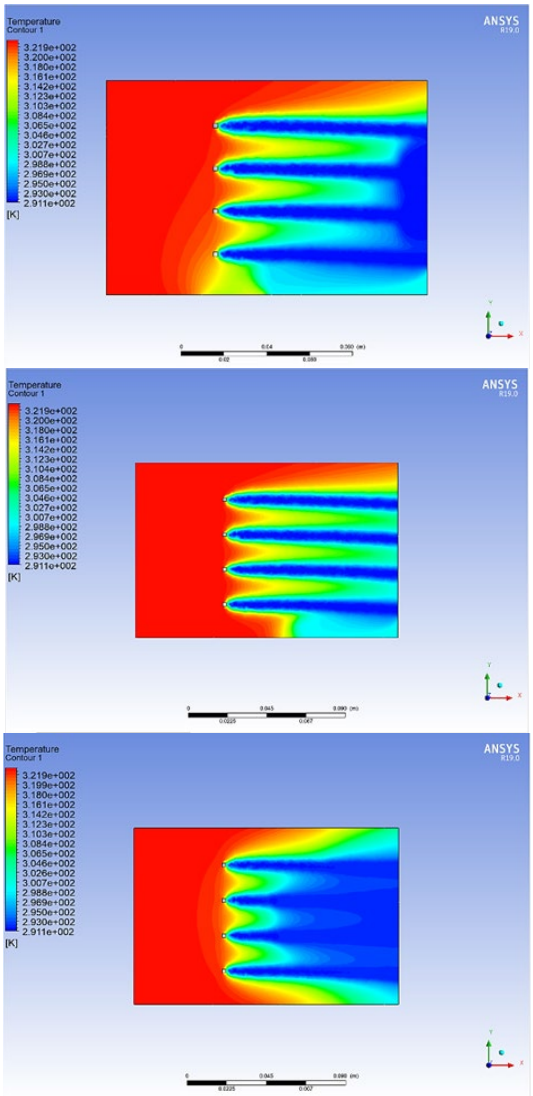
(a) Temperature



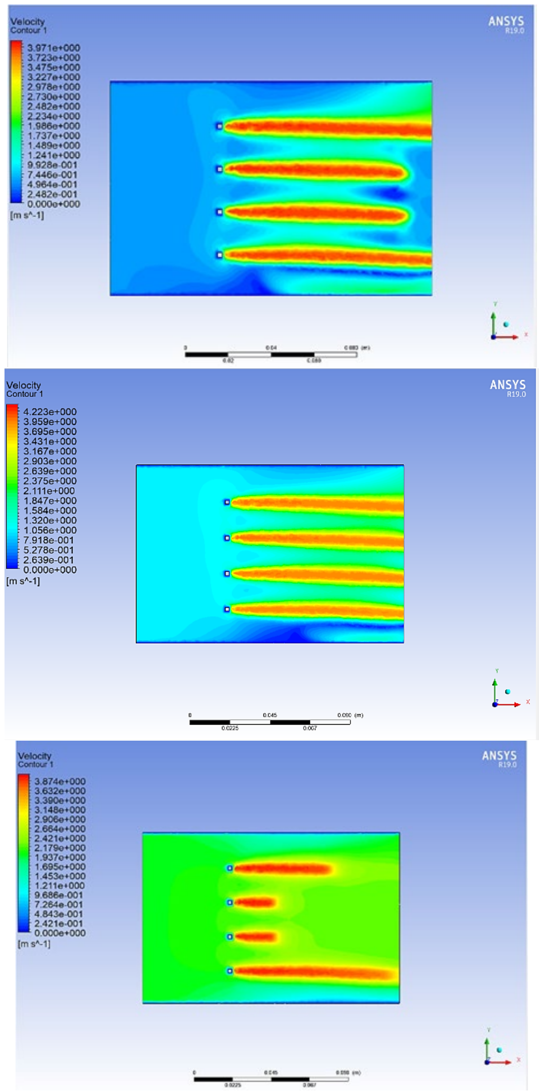


(b) Velocity

Figure 5. Temperature with contour of velocity for 15 nozzles with air velocities 0.5, 1.0, and 2.0 m/s

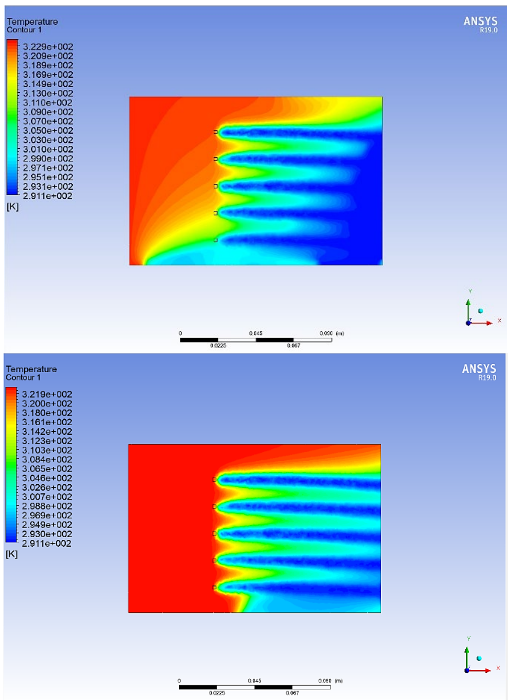


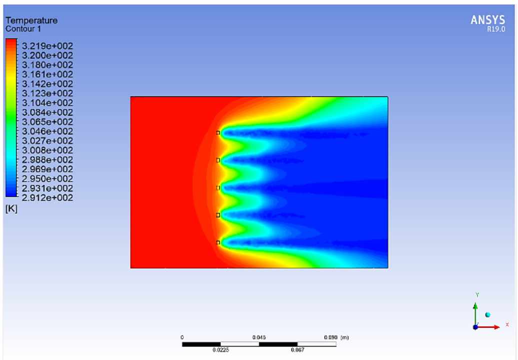
(a) Temperature



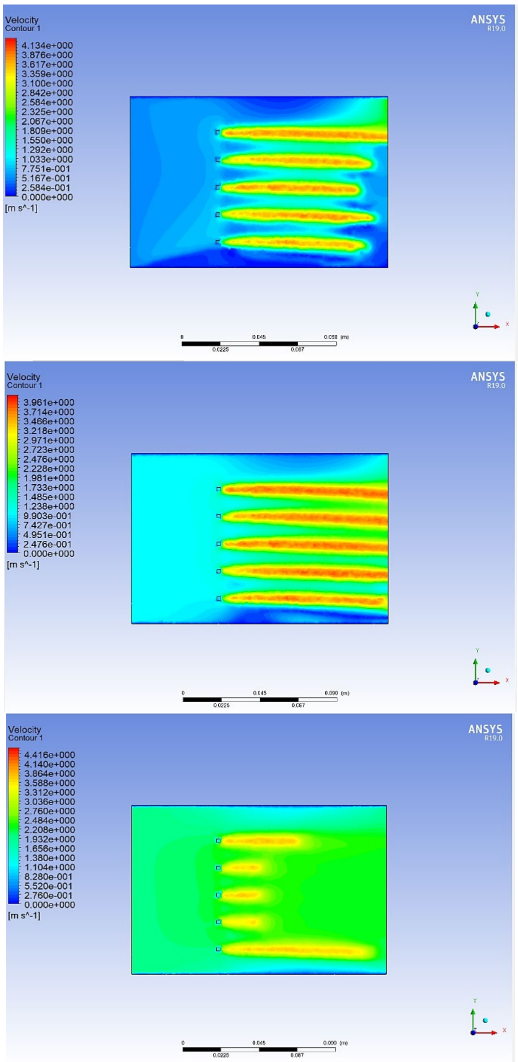
(b) Velocity

Figure 6. Temperature with contour of velocity for 24 nozzles with air velocities 0.5, 1.0, and 2.0 m/s





(a) Temperature



(b) Velocity

Figure 7. Temperature with contour of velocity for 35 nozzles with air velocities 0.5, 1.0, and 2.0 m/s

4.3 System efficiency assessment: Coefficient of performance analysis

The data in Table 2 show the effect of different quantities of injectors and the number of coils on the COP of the system under the influence of varying velocities of the incoming. Notably, the system is most efficient, as demonstrated by a coefficient of performance (COP) of 4.537, when the incoming speed is set to 2 m per second and 35 injectors and 13 coil turns are used. However, increasing the number of coils and injectors does not always result in an improvement in the COP. This finding is demonstrated by the minor decrease in COP when the 15 and 24 injectors are utilized with an increased number of rotations while maintaining the incoming speed at 1 m per second. This probable phenomenon can be attributed to the pressure drop induced by increasing revolutions, which can impede efficiency increases. Remarkably, the COP shows a rise regardless of the number of coil turns.

The COP is computed using 15, 24, and 35 injectors in the same number of coil turns. As shown in the table below, the best-performing injector is 35, which indicates that the three turns have higher values than the other turns.

As it is evident from Figure 8, the maximum COP is found to be almost equal to 4. 54 after 13 turns, whereas it increases to more than 4. 35 after 9 turns. Thirteen, nine, and five turns are quite distinct from each other; as the number of turns increases the coil rotates at a high turn rate thus increasing COP.

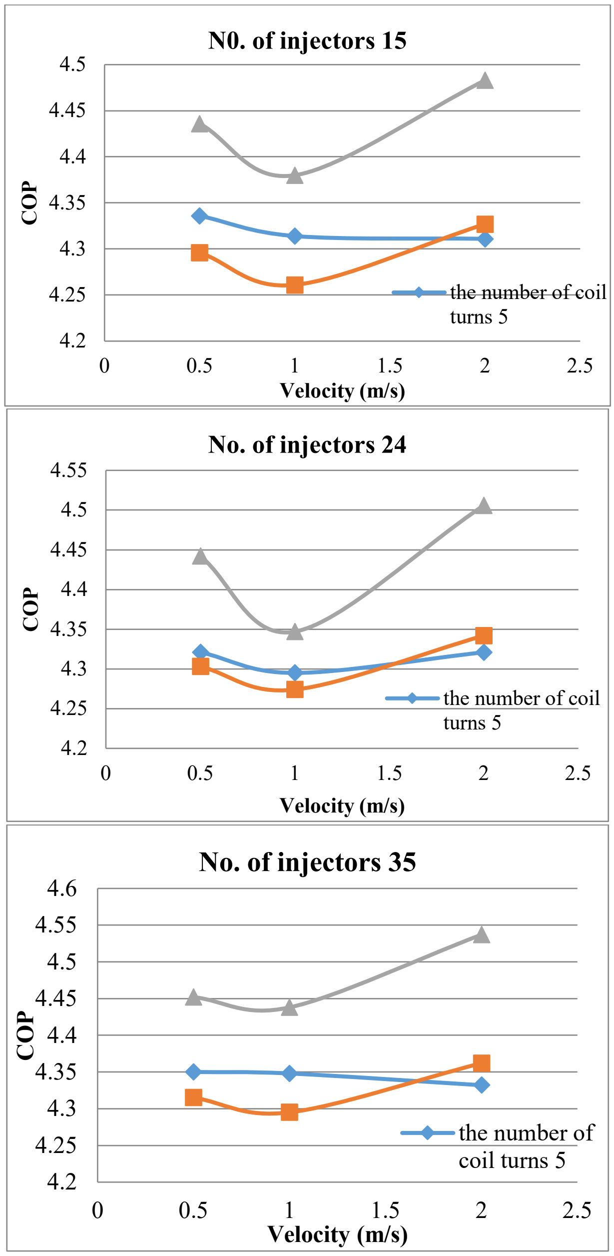


Figure 8. COP and number of turns

Table 2. COP for different input velocities, injector, and coil turn configurations

Inlet Velocity	No. of Injectors / Coil Turns	15/5	15/9	15/13	24/5	24/9	24/13	35/5	35/9	35/13
0.5		4.336	4.296	4.436	4.321	4.303	4.442	4.35	4.315	4.452
1.0	COP	4.314	4.261	4.38	4.295	4.274	4.347	4.348	4.295	4.438
2.0		4.311	4.327	4.483	4.321	4.342	4.506	4.332	4.362	4.537

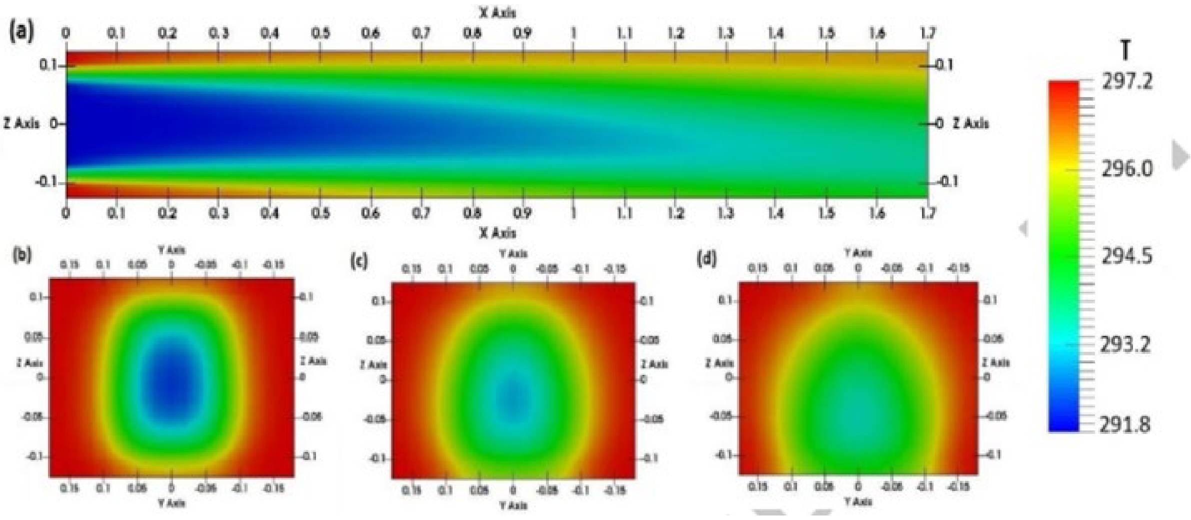


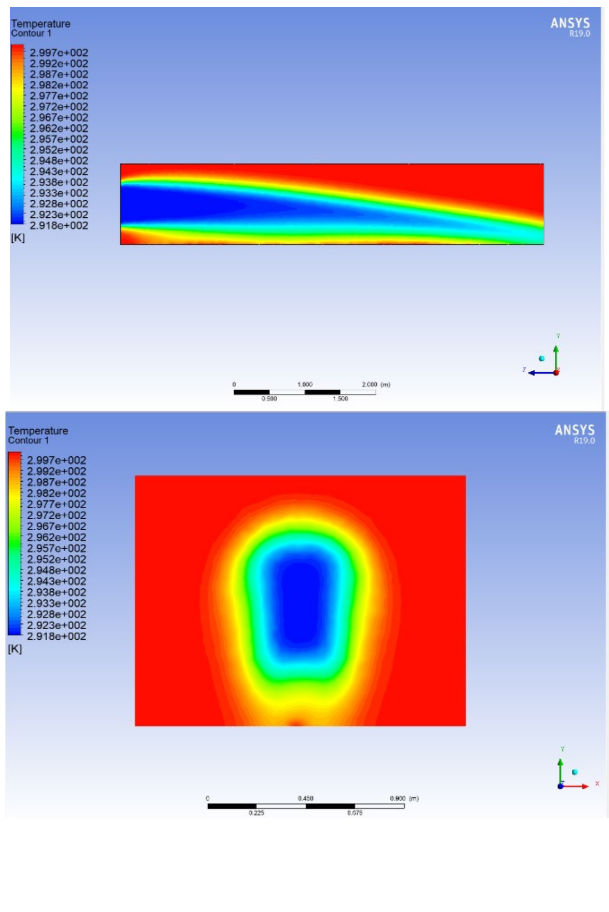
Figure 9. Temperature of air at y=0 m, x=0.5 m, x=1 m and x=1.5 m [18]

4.4 Validation

This shortcoming arises from numerous physical factors that make it difficult to study polydisperse evaporating spray. Research on cooling performance, however, has not considered the effects of water spray systems on heat exchangers, although some studies adopted CFD models. Recreating polydisperse evaporating sprays over complex 3D geometries is of great interest to the industry and has been made convenient and easy by the present work. This research is unique in that it is the first work that uses a CFD numerical tool to investigate the impact of water spray on heat exchangers and offers a CFD water spray model. The spray model has two stages: spray creation and airflow dispersion. The spray generation step takes place between the droplet injection and the air velocity. The droplet trajectory analysis gives this position and the spray size, the droplet size reduction equation gives the amount of the evaporated liquid water. The second component of the 3D CFD program Code_Saturne is the boundary conditions for this first segment of the program. The k-ϵ turbulence model is used in solving Navier–Stokes equations for the spray in this CFD code. The three transit variables are potential temperature based on liquid volume, L; total water mixing ratio including vapor, qw; and droplet concentration, Nc. Droplet evaporation is incorporated into the NC calculation by adopting the source-term approach. The use of the lognormal law for describing droplet spectra and their dynamics is explained.

A good correlation was observed when comparing Raoult et al.'s work [18] (Figure 9) and this work (Figure 10).

In summary, the research findings indicate a strong correlation between the performance of a heat exchanger and its geometric characteristics, specifically the number of turns in the cooling tubes and the quantity of nozzles. The greatest efficiency was realised when the configuration of 35 nozzles and 13 turns was used, this corresponds to the results presented in the works of Dhamneya et al. [19] and Al-Juwayhel et al. [20]. The number of turns improved the heat transfer as noted by Zhao et al. [21]. Likewise, the distribution of the fluid is enhanced when more nozzles are used; in agreement with the work of Chiesa et al. [22]. Nevertheless, it is seen that when these parameters are changed, the complexity and pressure drops will be high, which is in accordance with the conclusion of Tewari et al. [23] and Chiesa et al. [24]. Thus, optimization is a balance between these factors, which should be achieved in the most effective and efficient manner.



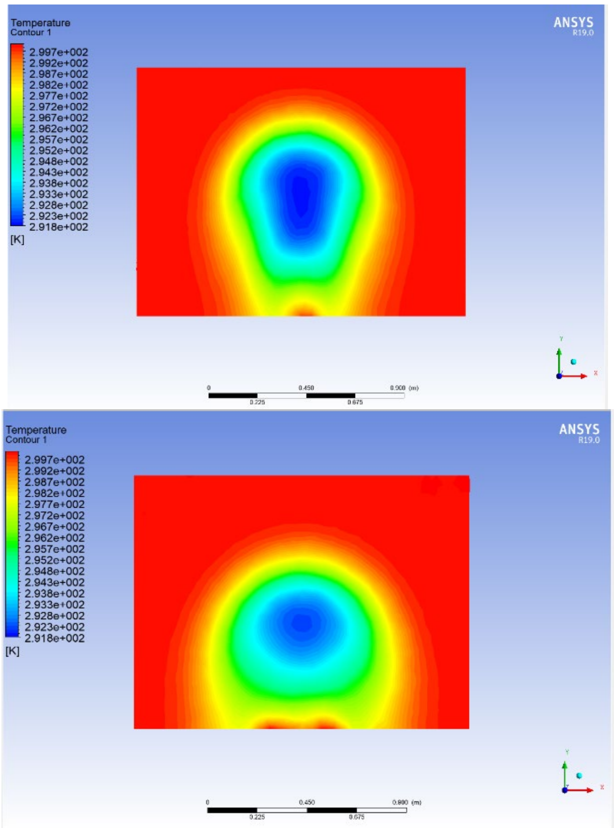


Figure 10. Temperature of air at y=0 m, x=0.5 m, x=1 m and x=1.5 m (present)

5. CONCLUSIONS

In this work, the influence of the geometrical parameters on the coil-and-tube heat exchangers for heat transfer was considered. Since the primary aim of this study was to establish the link between design factors and thermal performance, it used the conservation of mass, energy balance, and entropy balance equations. As it will be observed from the findings, system geometry has a significant impact on heat transfer performance. The total distribution of temperature and the efficiency of heat transfer by coils are highly sensitive to changes in the coil designs, the spacing of tubes, and the number of nozzles. It is also observed that more number of nozzles is better for the distribution of the fluid and increases the fluid velocity for better heat transfer rate. But to avoid any possible flow problems, the ratio has to be kept precise. Another factor that is required to be considered while deciding on the number of nozzles is the exact requirements of the heat exchanger application. The enhancement of the cooling tube turns increases the heat transmission because it increases the length of the heat exchange and turbulence. On the other hand, less number of tube turns is likely to reduce the flow path complexity thus reducing the demand for pumping power and pressure loss. These elements have to be properly adjusted to get the best setting of the configuration in question.

The following conclusions can be drawn:

1. The research points out the significant changes in the heat exchangers' efficiency due to geometric variables, with a focus on the number of nozzle and cooling tube bends.

2. Thus, using the enhanced simulation features of ANSYS, this work was able to explain the complex influences arising from system geometry on thermal behavior and fluid evolution.

3. As for the optimal configuration the system with 35 nozzles and 13 coil turns was considered to be the most effective. This configuration obtained the highest COP of 4. 537 when the inlet velocity is 2. 0.

4. The results extend the recognition of the ability of geometrical optimization to significantly increase the efficiency of heat exchangers.

5. The focus of this study is to demonstrate the effectiveness of computation in understanding and improving complex thermal systems.

6. The study was fruitful, but it also highlighted possible trade-offs; for example, pressure drop was found to be higher. This discovery is quite revealing, especially when it comes to the question of the balance between the components in the design of the system.

Thus, it should be noted that, despite the fact that our study provides useful information, it has certain limitations. Therefore, more experiments and refinement of the computational model is recommended to increase the prediction precision. Future research studies may analyze other features influencing the heat transfer efficiency and improve the suggestions for the suitable system design. Thus, extending the knowledge of the relationship between the geometry of the heat exchanging system and heat transfer efficiency in coil-and-tube heat exchangers, this work contributes to the development of the scientific discipline of heat transfer. These findings will form the basis of future advancements in the design and the enhancement of heat exchange systems.

The recommendations that may be made to improve the knowledge level on the effect of system geometry on heat exchanger performance are as follows as per the broad simulation analysis carried out with ANSYS. The current recommendations propose prospects for the outcomes and may be helpful for additional investigations and other objectives.

1. If the coil turn and nozzle count characteristics are studied in detail, this may be optimized and thus the system becomes more efficient.

2. The findings of the investigation might be extended to include the general impact of other fluids to the efficiency of the system under consideration.

3. It is necessary to study how the variation of the size of the duct influences the heat exchange process of the system.

REFERENCES

[1] Camargo, J.R., Ebinuma, C.D., Silveira, J.L. (2005). Experimental performance of a direct evaporative cooler operating during summer in a Brazilian city. International Journal of Refrigeration, 28(7): 1124-1132. https://doi.org/10.1016/j.ijrefrig.2004.12.011

[2] Eidan, A.A., Alwan, K.J., AlSahlani, A., Alfahham, M. (2017). Enhancement of the performance characteristics for air-conditioning system by using direct evaporative cooling in hot climates. Energy Procedia, 142: 3998-4003. https://doi.org/10.1016/j.egypro.2017.12.311

[3] Zhou, Y.Y., Yan, Z., Dai, Q.M., Yu, Y.S. (2021). Experimental and numerical evaluation of a two-stage indirect/thermoelectric assisted direct evaporative cooling system. Energy Convers Manag, 248: 114780.

[4] Mousavi, M.S., Mirfendereski, S., Park, J.S., Eun, J. (2019). Experimental and numerical analysis of a sustainable farming compartment with evaporative

cooling system. Processes, 7(11): 823. https://doi.org/10.3390/pr7110823
[5] Moshari, S., Heidarinejad, G. (2015). Numerical study of regenerative evaporative coolers for sub-wet bulb cooling with cross-and counter-flow configuration. Applied Thermal Engineering, 89: 669-683. https://doi.org/10.1016/j.applthermaleng.2015.06.046
[6] Adam, A., Han, D., He, W., Chen, J. (2021). Numerical analysis of cross-flow plate type indirect evaporative cooler: Modeling and parametric analysis. Applied Thermal Engineering, 185: 116379. https://doi.org/10.1016/j.applthermaleng.2020.116379
[7] Fan, B., Zhao, L.H., Zhang, Y., Sun, J.X., Xiong, W., Chen, J.Q., Chen, X.Z. (2020). Numerical study of evaporative cooling in the space station. Journal of Physics B: Atomic, Molecular and Optical Physics, 54(1): 015302. https://doi.org/10.1088/1361-6455/abc72d
[8] Jradi, M., Riffat, S. (2014). Experimental and numerical investigation of a dew-point cooling system for thermal comfort in buildings. Applied Energy, 132: 524-535. https://doi.org/10.1016/j.apenergy.2014.07.040
[9] Cui, X., Chua, K.J., Yang, W.M. (2014). Numerical simulation of a novel energy-efficient dew-point evaporative air cooler. Applied Energy, 136: 979-988. https://doi.org/10.1016/j.apenergy.2014.04.040
[10] Zhou, Y.Y., Zhang, T., Wang, F., Yu, Y.H. (2020). Numerical study and optimization of a combined thermoelectric assisted indirect evaporative cooling system. Journal of Thermal Science, 29: 1345-1354. https://doi.org/10.1007/s11630-020-1362-7
[11] Khafaji, H.Q., Ekaid, A.L., Terekhov, V.I. (2015). A numerical study of direct evaporative air cooler by forced laminar convection between parallel-plates channel with wetted walls. Journal of Engineering Thermophysics, 24(2): 113-122. https://doi.org/10.1134/S1810232815020022
[12] Riangvilaikul, B., Kumar, S. (2010). Numerical study of a novel dew point evaporative cooling system. Energy and Buildings, 42(11): 2241-2250. https://doi.org/10.1016/j.enbuild.2010.07.020
[13] Sheng, C.G., Nnanna, A.G.A. (2011). Empirical correlation of cooling efficiency and transport phenomena of direct evaporative cooler. Applied Thermal Engineering, 40: 48-55. https://doi.org/10.1016/j.applthermaleng.2012.01.052
[14] Majdi, H.S., Hasan, W.K., Hussein, M.A.M., Habeeb, L.J. (2023). Numerical implementation of direct and reverse flow for plate heat exchanger. Mathematical Modelling of Engineering Problems, 10(2): 491-500. https://doi.org/10.18280/mmep.100215
[15] Banooni, S.M., Habeeb, L.J., Hasan, A.H. (2023). Enhanced performance of structurally optimized plate-fin heat exchangers through numerical modeling of heat transfer and pressure drop. Journal of Sustainability for Energy, 2(2): 39-49. https://doi.org/10.56578/jse020201
[16] Hameed Hasan, A., Mehrzad Banooni, S., Habeeb, L. (2024). Heat transfer and pressure drop in plate fin heat exchangers: A numerical approach. Al-Rafidain Journal of Engineering Sciences, 2(1): 119-135. https://doi.org/10.61268/qe48kc51
[17] Saleem, J., Chahrour, K.M.N. (2024). Enhancing the efficiency of air conditioning systems in high-temperature climates through direct evaporative cooling. Journal of Sustainability for Energy, 3(2): 119-138. https://doi.org/10.56578/jse030205
[18] Raoult, F., Lacour, S., Carissimo, B., Trinquet, F., Delahaye, A., Fournaison, L. (2019). CFD water spray model development and physical parameter study on the evaporative cooling. Applied Thermal Engineering, 149: 960-974. https://doi.org/10.1016/j.applthermaleng.2018.12.063
[19] Dhamneya, A.K., Rajput, S.P.S., Singh, A. (2018). Thermodynamic performance analysis of direct evaporative cooling system for increased heat and mass transfer area. Ain Shams Engineering Journal, 9(4): 2951-2960. https://doi.org/10.1016/j.asej.2017.09.008
[20] Al-Juwayhel, F., El-Dessouky, H., Ettouney, H., Al-Qattan, M. (2004). Experimental evaluation of one, two, and three stage evaporative cooling systems. Heat Transfer Engineering, 25(6): 72-86. https://doi.org/10.1080/01457630490486292
[21] Zhao, R., Liu, J., Gu, J.J., Zhai, L., Ma, F. (2020). Experimental study of a direct evaporative cooling approach for Li-ion battery thermal management. International Journal of Energy Research, 44(8): 6660-6673. https://doi.org/10.1002/er.5402
[22] Chiesa, G., Huberman, N., Pearlmutter, D. (2019). Geo-climatic potential of direct evaporative cooling in the Mediterranean Region: A comparison of key performance indicators. Building and Environment, 151: 318-337. https://doi.org/10.1016/j.buildenv.2019.01.059
[23] Tewari, P., Mathur, S., Mathur, J. (2019). Thermal performance prediction of office buildings using direct evaporative cooling systems in the composite climate of India. Building and Environment, 157: 64-78. https://doi.org/10.1016/j.buildenv.2019.04.044
[24] Chiesa, G., Huberman, N., Pearlmutter, D., Grosso, M. (2017). Summer discomfort reduction by direct evaporative cooling in Southern Mediterranean areas. Energy Procedia, 111: 588-598. https://doi.org/10.1016/j.egypro.2017.03.221