



High-Performance Cooling: The Influence of Turbulent Air Inlet Velocity on Triangle Fins for Efficient Heat Transfer

Basma Abdulhadi Badday^{1*}, Lina Jassim², Muna S. Kassim³, Muhammad Asmail Eleiwi⁴, Hasan Shakir Majdi⁵

¹ Fuel and Energy Techniques Engineering Department, Technical Engineering College – Baghdad, Middle Technical University, Baghdad 10074, Iraq

² Mechanical Engineering Department, Mustansiriyah University, Baghdad 10052, Iraq

³ National Investment Commission, Mechanical Engineering Department, Mustansiriyah University, Baghdad 10052, Iraq

⁴ Electromechanical Engineering Department, College of Engineering, University of Samarra, Samarra 34010, Iraq

⁵ Department of Chemical Engineering and Petroleum Industries, Al-Mustaqbal University College, Hillah 51001, Iraq

Corresponding Author Email: basma.baddi@mtu.edu.iq

Copyright: ©2025 The authors. This article is published by IETA and is licensed under the CC BY 4.0 license (<http://creativecommons.org/licenses/by/4.0/>).

<https://doi.org/10.18280/ijht.430516>

ABSTRACT

Received: 12 July 2025

Revised: 6 October 2025

Accepted: 17 October 2025

Available online: 31 October 2025

Keywords:

triangular fins, turbulent flow, computational fluid dynamics, temperature distribution, heat dissipation efficiency

This work's results proved that by increasing air inlet speed from 2 m/s to 6 m/s, the maximum heat sink temperature decreased from 79.8°C to 52.5°C, avoiding hotspots, leading to an even temperature distribution. This study shows how using triangular fins mixed with higher turbulent air speed achieves considerably better temperature control for electronics cooling than alternate fin geometries or slower airflow. When used in a real-world application such as compact electronics or high-power devices, the combination of a single high-speed air inlet and triangular fin array allows for safer operating temperatures, longer device lifespan, and potentially allows greater power density without thermal hazards. Increasing air speed from 2 m/s to 6 m/s lowers the hottest location on the heat sink from 79.8°C to 52.5°C and provides much better temperature regularity from one end of the sink to the other. Higher turbulence (from both fin geometry and airflow speed) efficiently disrupts thermal boundary layers, which moves heat more quickly, preventing thermal buildup. This is because a single-velocity high-pressure inlet allows for better overall cooling than other methods of temporally multiplexing flow into multiple lower-velocity directional paths, especially relevant for electronic systems in which spatial compaction and efficiency are highly important.

1. INTRODUCTION

Good thermal management in electronics keeps hardware alive and ensures that it functions at its best. Extremely useful heat sinks play a part in accomplishing this goal. They protect components by removing excess heat, ensuring that everything continues to function at its optimal level. Because modern electronics are more compact and powerful than ever before, heat sinks are essential to ensuring that they continue to function securely and dependably in industries such as computing, transportation, and renewable energy.

Soloveva et al. [1] Used numerical methods on plate-fin heat sinks, changing fin counts and spacing, all while preserving the original shape to reduce manufacturing difficulties. They found that such adjustments help the heat sink be rid of heat up to 24.44% faster, use less metal, and become smaller by 30%. SST k- ω turbulence model was applied in computational fluid dynamics to give useful ideas for designing efficient and cost-effective microelectronic cooling. Khor et al. [2] used computational fluid dynamics to investigate the effect of the various inlet air and heating conditions on the cooling performance of circular pin fin, plate

fin, and rectangular fin heat sinks. Their results have shown that with the increase of inlet air velocity, the heat transfer rate is enhanced and maximum temperature values are lowered; the circular pin fin showed higher performance due to the large free stream openings. Ranjan et al. [3] also used CFD to study the thermal behavior of perforated hollow fins of circular, oval, and rectangular shapes. The results showed that rectangular perforated fins did offer the greatest cooling efficiency (types with a more complex manufacturing process).

Mirapalli and Kishore [4] reported that higher heat transfer efficiencies are seen for rectangular fins, but triangular fins are lighter and give a more uniform distribution of the temperature. Their results show that air-cooled engines must achieve the optimal balance between fin geometry, heat transfer capacity, and material requirements to achieve efficient fin designs. In 2023, Arslan and Pulat [5] used computational fluid dynamics to study the thermal response of a printed circuit board fitted with 4 heat generating chips under various air inlet velocities and heat flux conditions. Their simulations showed that the greater airflow over the board reduced the temperatures in the chips, but higher heat fluxes

presented more problems with cooling the electronic components. Huang et al. [6] examined the improvement of heat transfer efficiency in heat-sink fins by using synthetic jets later directed to the fins using a combination of experimental measurements and numerical simulations. This analysis quantitatively established that the synthetic jet induced turbulent boundary layer greatly lowered the surface temperatures on the fins and therefore supplied the key information for the heat sink geometry optimization in cooling systems of electronic apparatus.

Hempijid and Kittichaikarn [7] investigated the effect of inlet and outlet operation on heat transfer performance of a micro pin-fin heat-sink. Through an organized investigation of various inclinations, they found that heat performance was enhanced the most for a 75° inclination of the V-type heat sinks due to the strong vortex structures, which encourage effective mixing of heated fluids and increase heat dissipation. Adhikari et al. [8] studied the forced convection heat transfer through the rectangular straight fins at low Reynolds number using experimental and numerical methods, and checked the unsteady flow. The results revealed that there is almost a linear relationship between Nusselt and Reynolds numbers and that shorter channels are the factors responsible for improved effective heat transfer, as flow development is minimal there. Qian et al. [9] have performed three-dimensional CFD simulations to determine the effects of the inlet velocity profiles and changes of important geometrical parameters such as the fin length, fin angle, and fin pitch on the louver fin performance over the Reynolds numbers of 70 to 350. The results showed that the air entering arrangement of the heat exchanger has a large effect on its efficiency and pressure losses; hence, the best arrangement reduced both losses by around 19%.

This work by Pourdel et al. [10], through numerical simulation, considered flow and heat transfer in flat tubes with internal dimples, exploring their impact on both pitch and depth while the Re numbers were 5,000 and 20,000. What they found was that dimples affect the boundary layer, greatly improving heat transfer, but also increasing friction, so improving one aspect comes at the cost of the other. Hwang et al. [11] looked at how periodically dimple-protrusion patterned walls enhance the randomness of heat flux and temperature in small heat exchangers. It was found that at lower Reynolds numbers, and because of reciprocal vortices, heat transfer is improved more. Researchers Zhao et al. [12] relied on a numerical analysis to improve cooling channels with staggered elliptic dimples. As a result, two configurations were built where heat transfer increased by 32.8% and the pressure drop decreased by 34.6%. Rao et al. [13] looked at the difference between using a smooth and a dimpled air-cooling channel with pin fins. Using numbers and experiments, they found that vortexes created by dimples close to the pins boosted the amount of flow that played a role in boosting heat transfer. Kumar et al. [14] investigated a heating triangular fin array in a vertically oriented rectangular enclosure with air filling, to analyze how several parameters influence the system for a wide range: Rayleigh number $295214 \leq Ra \leq 773410$, fin spacing $25 \text{ mm} \leq S \leq 100 \text{ mm}$, fin height $12.5 \text{ mm} \leq L \leq 37.5 \text{ mm}$, assuming the heat flux to be constant at the enclosure's hot and cold boundaries. Khaled and Gari [15] analyzed the transfer of heat through a wall with triangular fins partially set inside its volume. Coupled equations for heat were solved using a numerical iterative finite volume method. Both numerical and analytical

outcomes are included in the authors' report. It has been found that the heat in the fin-root can be passed through to or from the wall in either direction. The combined system achieves the highest heat transfer rate at a certain fin-root length. High mechanical strength structures in contact with flowing fluids can benefit from the enhanced heat transfer offered by triangular rooted-fins.

Table 1. Three fin types comparison

Fin Type	Pros	Cons
Rectangular	High overall heat transfer, easy to manufacture, traditional design	Can be heavy, risk of hot spots, less turbulence, complex with perforation
Circular	Good airflow, lower flow resistance, uniform cooling at high velocities	Lower surface area contact, less turbulence, potentially lower thermal performance
Triangular	Enhanced turbulence, high heat transfer at high flow, light, uniform temperature	Can increase resistance, may be harder to manufacture, and have fewer benefits at low velocities

Each configuration of fins, ranging from rectangles, circles, and triangles, has its own disadvantages and advantages in heat sink thermal management, especially in electronics cooling. Based on reference studies and supporting literature, important points made about various fin types can be summarized as follows: the rectangular fins have been widely applied for simple geometry and ease of manufacture. They offer a large surface area for the dissipation of heat and promote the convective transfer of heat. According to the above-mentioned study, the use of a perforated, hollow, or hollow-perforated rectangular fin is recommended in such cases for better thermal performance. However, for their fabrication, perforated devices are more complex; the devices frequently display increased weight and surface area while having increased material consumption, which may limit their applicability, miniaturization capability, and the flexibility of their functions. The performance characteristics of these fins are further enhanced by having efficient overall heat transfer rates. However, they are also less uniform in temperature distribution, and possible local heating hot spots can be present in rectangular fins. Circular pin fins are more efficient for airflow because they are open between the pins and lower the resistance of the wall to airflow, preventing an increase in resistance and allowing for higher inlet velocities to help provide better cooling. In certain cases, such fins are beneficial and, where uniform fluid access to surfaces is important, even stronger. The research literature review proves that circular fins tend to work better with higher airflow velocities, frequently dropping maximum temperatures faster than rectangular configurations. However, for larger surface area contact, their energy transfer can be slightly inferior, and also more complex in terms of turbulent mixing. It can sometimes not be obtained as effectively as triangular profiles. Specifically, triangular fins utilized in this work offer specific advantages regarding heat dissipation (particularly in turbulent air flow). Higher surface area per unit mass than other geometries result in lighter designs with better thermal performance. More even temperature distribution on the heat sink, eliminating the hot spot hazard. Simulations (inlet and thermal data analyses) and literature evidence provide good evidence that triangular fins enhance convective heat transfer

more significantly than other types. The benefits might be lower for low air velocities or lower airflow design performance. The key issues are difficulties in manufacturing (for instance, for small or complex triangular arrays) and pressure drop (increased turbulence can, in turn, make a surface more resistant to airflow). Table 1 summarizes the differences between the three types according to the above research papers.

2. SIMULATION MODEL SETUP AND BOUNDARY CONDITIONS

The evaluation of heat transfer from a heat source located beneath a linear triangular fin on a horizontal surface was conducted in an air tunnel, as seen schematically in Figure 1(a). The setup comprises an air tunnel, a heat sink positioned within the test section of the air tunnel, and an electrical heating element attached to the base of the heat sink. The air tunnel walls are located 25 mm from the edges of the heat sink in all directions, except at the outlet, which is 150 mm from the back edge of the sink, and no space between the heat sink bottom and the air tunnel bottom as shown in Figure 1(b). The uniform velocity, U_{∞} , was modified from 2 m/s to 6 m/s in increments of 2 m/s, maintaining a turbulence intensity of 0.5%. The constructed air channels and heat sink mimic fin channels oriented with the incoming flow. A heating element was attached to the base of the heat sink, which was considered thermally insulated to reduce heat losses. The heat flux applied to the lower surface of the heat sink was $10,000 \text{ watts/m}^2$. The steady-state temperature of the fin surfaces, T_s , was measured at $U_{\infty} = 2, 4$, and 6 m/s, the last three models with the three-inlet velocity had just one inlet as shown in Figure 1(a), the fourth case has a 2 m/s inlet velocity with three inlets from the front and two sides. The temperature was measured at four separate locations on the fin surface at the same height from the base to evaluate consistency. The study investigated the results of the inlet velocity variations in the first three cases, then compared the best results with the three inlet models with the lowest inlet velocity (2 m/s) to find the optimum design.

where, A ($= sH$) represents the channel cross-sectional area, and P ($= s + 2H$) indicates the wetted perimeter of the channel. The average Nusselt number (Nu) is computed utilizing the average forced convection coefficient (h) and hydraulic diameter (D_h) as follows [16]:

$$N_u = hD_h/K_f = QD_h/(A_t(T_s - T_\infty)K_f) \quad (2)$$

where, k_f signifies the thermal conductivity of air at the film temperature, A_f represents the total convective heat transfer surface area of the fins, T_s denotes the fin surface temperature, and T_∞ indicates the free-stream inlet air temperature. All air properties are evaluated at the film temperature $T_f = (T_s + T_\infty) / 2$, and variations in air properties are deemed unimportant in the numerical simulations due to the small temperature rise.

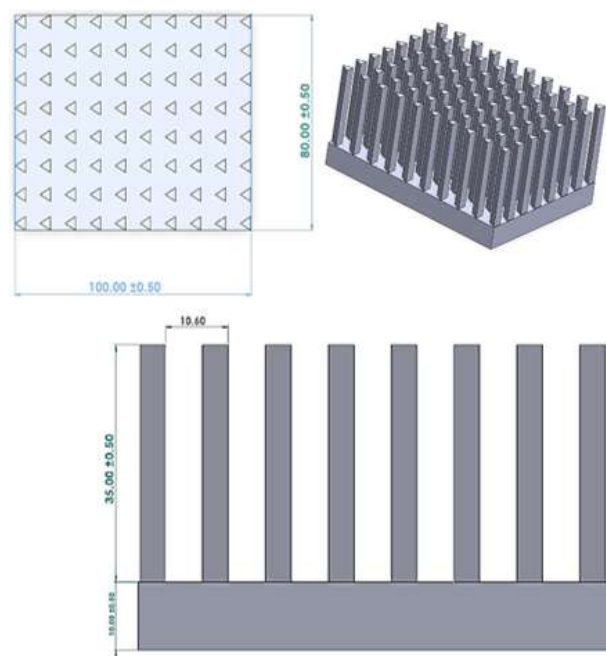


Figure 2. Heatsink SOLIDWORK model design and dimensions

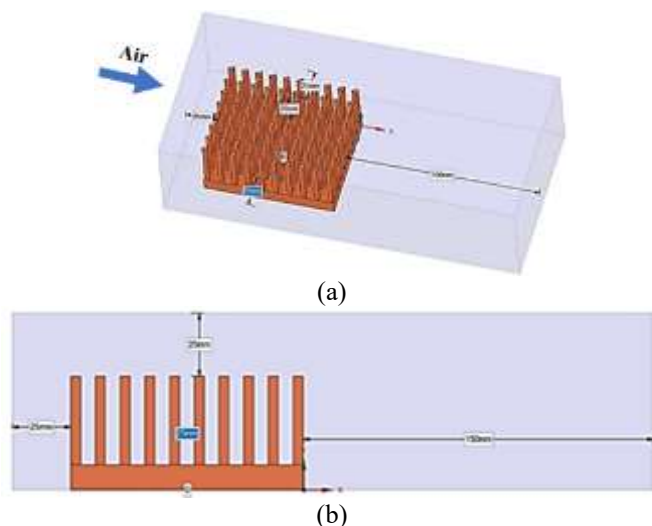


Figure 1. (a) Air tunnel (enclosure) dimensions from the heat sink, (b) bottom side dimensions

The hydraulic diameter of the channel section is defined by:

$$D_h = 4A/P \quad (1)$$

The fin array analyzed in this research is representative of those employed in the cooling of electronic devices with a fan. The SOLIDWORKS model and fins dimensions are shown in Figure 2. The fins were constructed from aluminum alloy. In the numerical calculations, it was assumed that the thermal conductivity, $K = 167 \text{ W/mK}$, and the emissivity ≈ 0.8 . The air temperature at the fin array entrance was 20°C . Reynolds number ranges from 2692 to 6728, signifying a turbulent flow regime. The heatsink model was designed by the SOLIDWORKS software, then imported to the ANSYS fluid, and the dimensions of the heatsink model are illustrated in Figure 2. Researchers examined heat transmission and flow dynamics using three-dimensional flow simulations using ANSYS Fluent CFD. Radiative heat transfer was modeled utilizing the surface-to-surface radiation model from sources [17] for triangular straight fins in natural convection; further details are available in the previous study [18]. The surface-to-surface radiation model evaluates energy exchange between fin surfaces, determined by their dimensions, separation, and orientation, quantified by a geometric quantity known as the view factor. Additionally, it presumes that absorption, emission, or scattering may be disregarded. Consequently, the radiation energy flow from the fin surfaces comprises emitted

and reflected energy. The characteristics of the intake air utilized in the simulations were as follows: the inlet air density was treated as an incompressible ideal gas, specific heat capacity (C_p) was 1006.34 J/kgK, viscosity was modeled using Sutherland's law, thermal conductivity (k) was 0.0242 W/mK, and the molecular weight was 28.966 kg/kmol. The turbulent flow was simulated using the realizable k- ϵ model coupled with standard wall functions, and a no-slip boundary condition was applied on the fin surface.

3. MESH INDEPENDENCE TEST

Ensuring that no changes or inconsistencies occur in simulation results with changing mesh densities is an important part of the standard protocol for ANSYS CFD simulations - mesh independence tests. The main objective is to find the best mesh density to maintain the computational efficiency and, at the same time, obtain a good solution. A series of simulations using a variety of mesh sizes, ranging from coarse to fine, is performed in order to determine significant variations in results when conducting a rigorous mesh independence assessment. We capture mesh independence, as indicated by the little variation in the results when the result is refined. The simulation used a mesh independence study consisting of ten different mesh sizes ranging from 0.5 mm to 5 mm in 0.5 mm steps. In this type of testing framework, the element size is the key determinant of the resolution of the mesh. The smaller the elements, the greater the number of mesh components that are generated in the analysis, and vice versa [19]. The temperature of the cold air was considered the main measurement parameter in the test. Table 2 lists the test results as a function of the mesh element size in terms of air tunnel outlet temperature, and it also shows comprehensive mesh information, including the number of corners and solid elements. In a CFD simulation, mesh independence verification is an imperative task to ensure the accuracy and reproducibility of the results. The test requires repeated grid refinement to verify that such quantities as temperature stay constant and do not fluctuate with continued grid densification. This way is used to identify the best mesh size between computation speed and accuracy of the results.

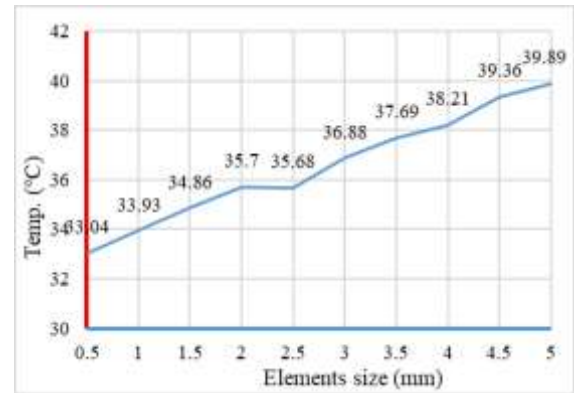


Figure 3. Mesh independence test results graphical chart

Figure 3 shows the temperature alteration with element size (mm) dependence in the computational domain. As the mesh element size gets smaller from 5 mm to 0.5 mm, the temperature readings approach a stable value. A significant rise in temperature is noted for mesh resolutions greater than 3 mm, which may indicate that mesh resolutions of this order of magnitude are a poor representation of the actual thermal gradients and patterns of flow, indicating that the limited spatial resolution of the model is responsible.

For element sizes between 2.0 mm and 2.5 mm, the temperature values are almost the same, with a temperature value of 35.70°C and 35.68°C for element sizes of 2.0 and 2.5 mm, respectively. This plateau means that the mesh has only become independent of the results, and further mesh refinements would be expected to involve little or no further change. When the element size is smaller than 2.0 mm, temperature fluctuations may occur; however, it takes considerably longer if the mesh is changed to smaller values. On the other hand, element dimensions greater than 2.5 mm give rise to inaccuracies in the predicted temperature.

The CFD simulation gives the best results between 2.0 mm and 2.5 mm of elements. This resolution of the mesh provides accuracy of the model without being computationally infeasible. Continue to use this mesh size for modeling unless a comprehensive examination is required. Consistently assess the mesh quality and ensure all components are appropriately correlated to maintain the reliability of the CFD findings. Figure 3 illustrates these results in a graphical chart.

Table 2. Mesh independence analysis of the simulation model

Element Size (mm)	Nodes No.	Elements No.	Corner Nodes	Solid Elements	Outlet Temp.°C
0.5	6144596	35121461	6144596	35121461	33.04
1	1094969	6149195	1094969	6149195	33.93
1.5	427882	2377516	427882	2377516	34.86
2	211899	1166588	211899	1166588	35.70
2.5	125894	689230	125894	689230	35.68
3	82797	449512	82797	449512	36.88
3.5	57888	312225	57888	312225	37.69
4	41012	220189	41012	220189	38.21
4.5	29159	154041	29159	154041	39.36
5	22393	117122	22393	117122	39.89

4. NUMERICAL METHOD

Case modeling and resolution using ANSYS FLUENT 2024 R1. A distinct implicit solver option was used to solve the governing equations. Mass, momentum, and scalar transport governing equations are discretized first, followed by

the formulation of an algebraic system of equations. Computational fluid dynamics (CFD) programs often use the finite volume technique, a branch of the finite difference numerical methodology, to compute flows. The fundamentals of finite volume computations will be covered here. The RANS equations were discretized using the k- ϵ

turbulence model throughout the instance executions of this project in order to account for flow variations caused by turbulence. In order to describe differential expressions in up air or other higher-order techniques, the equations were discretized using distinct differencing systems. The algebraic equations derived from the integral equations were solved at each cell node. These equations contained terms from the differencing scheme related to turbulence parameters, momentum, and energy [20]. In order to get the value of the scalar attributes, such as temperature, at a certain point in the computational domain, the calculation also has to consider the flow's direction and velocity. One sequential technique that outperforms the linked solver in terms of memory use is the segregated solver, which is used in numerical computation. The SIMPLE method is utilized for the computations in this project. Executed the standard procedure for pressure interpolation and SIMPLE pressure-velocity coupling. As seen in Figure 4, the CFD simulations, which comprised 20,000 iterations, were designed with a residual root-mean-square (RMS) goal value of 10⁻⁴ for continuity and 10⁻⁵ for the energy equation.

The realizable version of the k-ε turbulence model is the more popular one, which the authors select as a robust, computationally-effective approximation of fully turbulent flows in general, especially the ones relevant in practical engineering applications (e.g., heat sink air cooling applications). Therefore, airflow over triangular-shaped fins of an air tunnel in the current study has a Reynolds number in the range between 2,692 to 6,728, having dominant turbulence in the airflow, without an overpowering high-Re number causing it to exceed high-Re. The k-ε model is a strong weapon, especially when weighed against the typical wall functions to model the mean flow properties, level of turbulence, and heat transfer without significant amounts of computational expenditure. The k-ε model solves two transport equations: k, the transport equation of turbulent kinetic energy; and epsilon, the dissipation equation of kinetic energy. This provides a

good compromise between reality in physical terms and numerical stability. The authors used the so-called realizable model, which makes the flows with large recirculation and streamline curvature - that matter in the triangular fin designs with separation region and reattachment zone that have high heat transfer - more accurate. What is more, the k-ε model is highly applicable in ANSYS Fluent using a standard calibration constant set and has a high capacity in addressing forced convection cooling issues. Table 3 shows the essential differences between K-ε, K-ω, and (LES Large Eddy Simulation) models.

In this study, it is a forced convection problem, in which the flow is overall turbulent and steady. The main objective is to comprehend average heat transfer, but not transitional eddy processes. The k-ε model is the best compromise model between realism, numerical stability, and computational time. Other options, such as the SST k-ω, might be slightly more accurate with the wall layers, but more expensive, and LES would be overly expensive at design-level cooling simulations. Therefore, the selection of the authors is consistent with the conventional CFD approach to the turbulent cooling of electronic heat sinks.

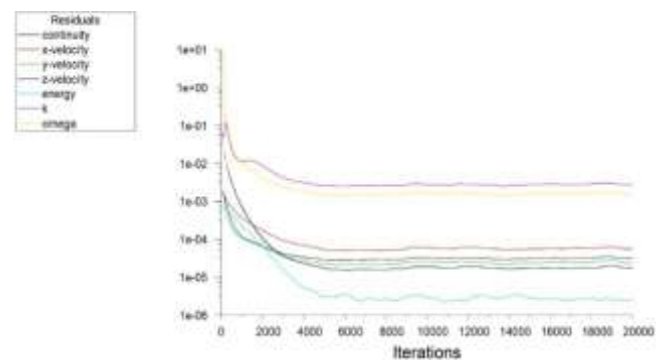


Figure 4. Simulation energy equation solution

Table 3. K-ε, K-ω, and (LES Large Eddy Simulation) models' comparison

Model	Strengths	Weaknesses	Suitability for this Study
k-ε (Realizable)	Reliable for fully turbulent internal/external flows; stable; low computational cost; good for heat transfer prediction in high-Re cases.	Less accurate near walls and in low-Re zones; averages out small-scale eddies.	Excellent balance for simulating high-speed air cooling of fins with limited CPU time.
k-ω (SST variant)	Better near-wall performance; useful for transitional and adverse-pressure-gradient flows.	More sensitive to inlet conditions; may overpredict turbulence away from walls.	Suitable but more computationally demanding for this geometry; less global stability for large domains.
LES (Large Eddy Simulation)	Captures full turbulence spectrum; highly accurate spatial resolution; resolves eddy effects directly.	Extremely high computational cost; requires a fine mesh and small time steps.	Impractical for steady cooling analysis with multiple fins and a large domain volume.

5. GOVERNING EQUATIONS

CFD is built on the Navier-Stokes equations, which detail the movement of a fluid. Incompressible flow can be described using these equations [21]:

$$\rho \left(\frac{\partial v}{\partial t} + (v \cdot \nabla) v \right) = -\nabla p + \mu \nabla^2 v + F \quad (3)$$

These equations mention: v for the vector velocity field, ρ for a constant fluid density (in case of incompressible flow), p for the pressure field, μ for dynamic viscosity, and F for the body forces acting on the fluid (e.g., gravity). This side of the equation measures the acceleration of the fluid, consisting of

the local change ($\Delta v / \Delta t$) and the transport effect ($(v \cdot \nabla) v$). On the right, you have the pressure gradient ($-\nabla p$), viscous forces ($\mu \nabla^2 v$), and outside forces (F). In steady-state flow, the rate of change with time disappears because $\partial v / \partial t$ is zero. Mass is still preserved in incompressible flows due to the continuity equation. In the case of an incompressible fluid (the density cannot change), ρ :

$$\nabla \cdot v = 0 \quad (4)$$

where,

$$\nabla v = \frac{\partial v_x}{\partial x} + \frac{\partial v_y}{\partial y} + \frac{\partial v_z}{\partial z} \quad (5)$$

As long as fluid obeys this equation, the mass of fluid is conserved, and the volume of fluid flowing in is the same as the volume flowing out. The energy equation is concerned with how temperature is spread within the fluid. The model includes convection as well as diffusion of energy within the fluid. A general form of the transient energy equation is:

$$\rho C_p \left(\frac{\partial T}{\partial t} + v \cdot \nabla T \right) = \nabla \cdot (K \nabla T) + \bar{q} \quad (6)$$

T : the temperature field, C_p : specific heat at constant pressure, k : thermal conductivity of the fluid, q : heat generation rate (per unit volume) coming from the heat flux source at the bottom of the heat sink. The rate that thermal energy changes in time appears on the left side ($\partial T / \partial t$), while the right side includes the transportation of thermal energy carried by the flow ($v \cdot \nabla T$). On the right-hand side are thermal diffusion ($\nabla \cdot (k \nabla T)$) and a heat source term q that is often the heat coming into the heat sink from its bottom. If there is a steady state, the time-dependent part of the derivative disappears from the equation. Heat flux is usually handled through a boundary condition at the top of the material (where heat leaves the heat flux source).

$$-K \nabla T \cdot \hat{n} = \bar{q} \quad (7)$$

where, k is the thermal conductivity of the material, ∇T is the temperature gradient normal to the surface, $\hat{n}$ is the unit normal vector to the surface of the heat sink. This boundary condition ensures that the heat flux q is applied to the heat sink's surface, representing the heat transfer from the source. Many engineers use the k- ϵ model to deal with turbulence in practical situations. There are just two main equations in this model, which are:

$$\frac{\partial K}{\partial t} + v \cdot \nabla K = \nabla \cdot (\mu_T \nabla K) + P_k - \epsilon \quad (8)$$

where, k is the turbulent kinetic energy, μ is the turbulent viscosity, P_k is the production term of turbulence, ϵ is the dissipation rate of turbulence. Dissipation Rate (ϵ) Equation:

$$\frac{\partial \epsilon}{\partial t} + v \cdot \nabla \epsilon = \nabla \cdot (\mu_T \nabla \epsilon) + C_1 \frac{\epsilon}{K} P_k - C_2 \frac{\epsilon^2}{K} \quad (9)$$

where, C_1 and C_2 are model constants, P_k is the production term (related to the strain rate of the flow). The turbulent viscosity μ_T is computed as:

$$\mu_T = \rho C_\mu \frac{K^2}{\epsilon} \quad (10)$$

6. RESULTS AND DISCUSSIONS

6.1 Heatsink temperature profile analysis

The researchers study the way temperature reaches different parts of the heat sink and compare the heat released in three heat sink models using air speeds of 2 m/s, 4 m/s, and 6 m/s. The simulations relied on ANSYS software, which is used for computational fluid dynamics (CFD) and thermal analysis. Aluminum is commonly used for making heat sinks because it conducts heat well and has a good price for its performance.

All three models are exposed to a steady supply of heat from below, representing what happens in actual heat sinks: the base creates heat that needs to be removed quickly and efficiently to the surrounding world. The heat distribution models and the outlet average temperature of the three simulation models with 2, 4, and 6 m/s inlet velocity are illustrated in Figures 5(A), (B), and (C).

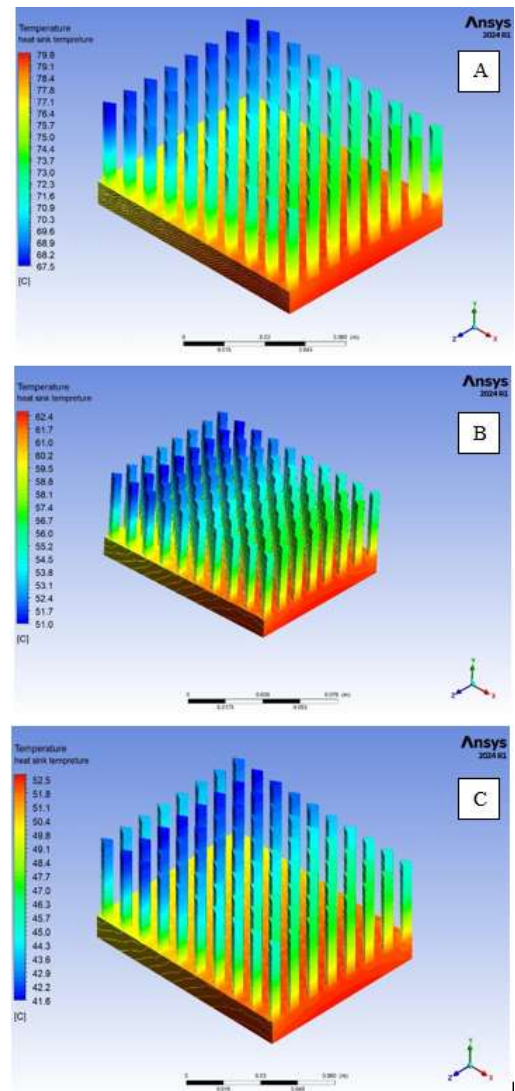


Figure 5. Heatsink temperature distributions, (A) 2 m/s inlet speed, (B) 4 m/s inlet speed, and (C) 6 m/s inlet speed

As seen in Figure 5(A), where the air speed at the inlet is just 2 m/s, the temperature distribution on the heat sink features a greater change in temperature. Air is weaker at the ground, so it does not cool things as much as expected. You can see from the thermal contours (Figure 5(A)) that the heat sinks coupling in this system have temperatures that vary from roughly 67.5°C to 79.8°C. According to the heat sink fins at the top, the cooling is less efficient, probably because a slow airflow makes it difficult for heat to move away. Very high temperatures are seen at the base, indicating that some heat is being trapped there, which becomes a worry for overheating if this system is used for practical purposes. An increase in air speed (to 4 m/s, in Figure 5(B)) clearly results in a faster transfer of heat from the heat sink to the air. By increasing the air speed to the continental case, the temperature decreased, temperature range between 51–62°C was recorded at this speed. The greater the air speed is, the faster heat is transferred,

simply because more air contacts the surface of the heat sink. The extra air moving helps the heat sink dispose of heat more effectively, mainly from its highest area. The effect is that the temperature is more even throughout the heat sink and less heat forms at the bottom. It is more effective thermally to run the system with an inlet speed of 4 m/s because it produces less heat and takes less effort to cool than when the speed is only 2 m/s.

Among all the settings, the strongest air inlet at 6 m/s (Figure 5(C)) is the best for cooling the object. The range of temperature in this area is lower; it drops from around 41.6°C to 52.5°C. More efficient heat transfer occurs because increased airflow speed improves the convective heat transfer coefficient. Because of this, the temperature at the base of the heat sink is now much lower than before in conditions when airflow is low. The fins of the heat sink are now colder, and the air temperature is more uniform, which means heat is being spread away efficiently. This setup gives the highest cooling performance for this kind of heat sink under the specific conditions. Fundamental principles of thermal and fluid dynamics are pivotal to the configurations of the observed temperature profiles. Increasing airflow velocity enhances the convective heat transfer coefficient, hence improving heat removal efficiency. Air traveling at 6 m/s conducts heat far more rapidly than air moving at 2 m/s. The acceleration of incoming air and expanded heat exchange area improve convective heat transfer, reducing temperature. Reduced airflow leads to a broader boundary layer, reducing heat transfer efficiency. Elevating airspeed reduces boundary layer thickness, enhancing heat transmission efficiency. Turbulence intensifies at a maximum airflow velocity of 6 m/s, enhancing fluid dynamics and heat dissipation. Aluminum heat sinks conduct heat from the base and disperse it along fins, affecting thermal efficacy. Fin configuration, quantity, and spacing also affect thermal efficacy. Elongating and thinning fins increase surface area for heat transfer but also affect thermoregulation based on air velocity [22].

6.2 Heatsink temperature distribution analysis

Figure 6 shows how the temperature is distributed on a heatsink center along the x-axis at 0.025 meters high from its base for air inlet rates of 2 m/s, 4 m/s, and 6 m/s. The length of the air tunnel in which the heatsink is placed goes from -0.1 to 0.20 meters on the x-axis, while the heatsink area lies within -0.10 and approximately 0.002 m. The graph illustrates how temperature changes as the air moves through the air inlet. As we move the heatsink at varying speeds (2 m/s, 4 m/s, and 6 m/s), we notice that it gets hotter in the middle before the temperature goes down as air leaves it. The area closest to the heat source or the center of the heatsink seems to get the hottest. With increased velocity, there is usually less peak temperature in the heatsink, which means the higher the airflow, the more effectively the heatsink is cooled. A higher velocity causes convection to speed up heat transfer, lowering the heat sink's hottest point compared to low velocities. Near airflow velocity of 2 m/s, a location near the center of the heatsink (about 0.002 m along the x-axis) may attain a temperature of 45°C. The spectrum of potential temperatures diminishes with an increase in velocity. Enhanced air circulation facilitates superior heat dissipation, with air reaching its peak temperature of around 35°C at a velocity of 4 m/s. At an air speed of 6 m/s, the temperature may attain 30°C. The heatsink appears to be cooled most efficiently by

the swiftest airflow. As one progresses laterally from the heatsink along the x-axis, air temperatures decrease, facilitating the transfer of heat from the heatsink to the air within the air tunnel. The majority of heating occurs near the heatsink, with a rapid decline in temperature as the distance from it increases, as seen by the temperature graphs for the three speeds. The heat sink's core, at 0.00 m, has the highest thermal intensity due to direct solar radiation heating. Heat transfers through convection, and as airspeed increases, efficiency improves.

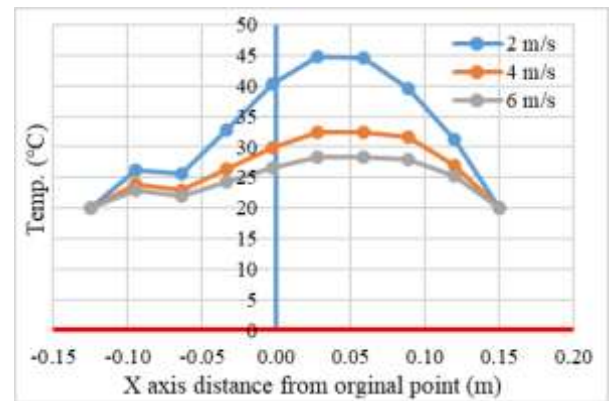


Figure 6. Heatsink temperature distribution along the x-axis at the heatsink center for the three inlet velocities

Air molecules directly touch the heatsink, removing thermal energy. However, 2 m/s air flow increases boundary layer thickness, reducing heat dissipation efficiency. Accelerated airflow decreases the opposing effect and therefore increases heat transfer. At a velocity of 2 m/s, the air flow forms a dense thermal boundary layer and which prevents the heat flow. As the air velocity gets even higher, the thickness of this boundary layer reduces further, and hence a better transfer of heat is achieved from the heatsink to the surrounding air. Heat is transferred into the heatsink, causing a slow reduction of its temperature. At higher air velocities, the convective heat dissipation becomes more efficient. Laminar flow is dominant for low velocities, while turbulent flow allows better mixing of air and thus improved heat transfer. Increased air velocity across the heatsink improves its heat dissipation capacity, reducing thermal resistance [23]. Figure 7 illustrates how the temperature varies near the heatsink outer wall (at a distance of 0.003 m) and along the x-axis, for three inlet velocities of 2 m/s, 4 m/s, and 6 m/s. On the x-axis, the length of the air tunnel and where the heatsink is located runs from -0.1 m to 0.002 m. The maximum recorded temperature of around 27°C was seen at the heat sink at a velocity of 2 m/s. It pertains to the area in contact with the heatsink, when thermal energy from the solid transfers to the air. As air moves down the positive x-axis, a rapid decrease in temperature is observed, particularly at greater distances from the heatsink. At an air speed of 4 m/s, the temperature reaches a maximum of around 23°C, which is lower than the top of 28°C seen at 2 m/s.

Thus, enhanced air movement improves the efficiency of convective heat transfer and the removal of heat. The reduced temperature differential allows air traveling at 5 m/s to collect and dissipate heat from the underlying ground more efficiently. The minimum temperature occurs at a velocity of 6 m/s, reaching around 22°C. This demonstrates an efficient method for dissipating heat. In this scenario, the gradient is less pronounced, indicating that the temperature decreases at a

more uniform pace within the heatsink. As the air accelerates, heat dissipates more effectively from the sink.

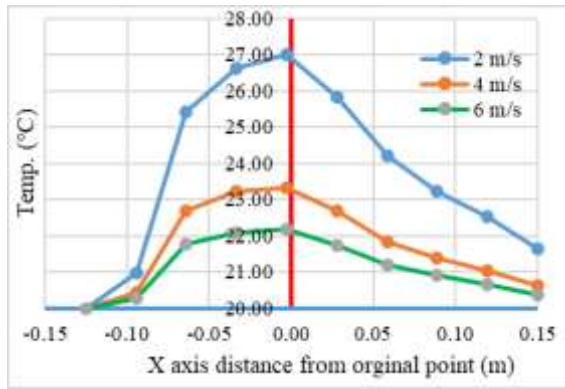


Figure 7. The distribution of the heatsink temperature along the x-axis at the heatsink wall for the three inlet velocities

The air near the heatsink wall ($x = 0$ m) exhibits the maximum temperature due to its proximity to the heater. As air velocity increases, peak temperatures decrease (from 27°C to 22°C). Moving right along the x-axis (away from the heatsink): As air advances, temperatures decrease, with higher speeds indicating better heat dissipation along the airflow. Low air velocity leads to a larger boundary layer near the heatsink wall, reducing heat transfer efficiency. At 4 m/s, thermal diffusion occurs more easily due to surface movement. At 4 m/s, the boundary layer diminishes, allowing efficient heat transfer across surfaces. The temperature around the heatsink decreases, allowing downstream air to absorb and dissipate heat, facilitating swift cooling. An increase in velocity increases the heat transfer coefficient, facilitating more effective heat removal. At 2 m/s, the boundary layer is extensive, resulting in elevated surface temperatures and increased thermal resistance. At 4 m/s and 6 m/s, heat transmission is more efficient due to reduced boundary layer thickness, allowing heat to dissipate more efficiently and securely. Laminar flow transpires at reduced velocities, resulting in prolonged heat dissipation [24]. Turbulent flow is more prevalent at speeds of 4 m/s or 6 m/s compared to lesser velocities of 1 m/s and 2 m/s. In turbulent heat transfer, the flow becomes chaotic, enhancing the mixing rate and reducing the thickness of the fluid layer where heat absorption occurs. Aerial heat transfer accelerates heat removal from the heatsink as air movement increases, hence reducing the temperature differential between the air and the heatsink surface. The data indicate that temperatures around the heatsink decrease as air velocity increases.

6.3 Air tunnel bottom temperature profile analysis

The heat sink's fins in Figures 8(A), (B), and (C) show thermal energy moving out into the open air tunnel. The plotted curves indicate the difference in temperature and the way that temperatures are arranged behind the heat sink while air flows in the tunnel. Models are tested with the base temperature set, and each design has air moving into the model at 2 m/s, 4 m/s, and 6 m/s.

In Figure 8(A), a significant amount of heat accumulates behind the heat sink fins when air flows at 2 m/s, as indicated by the elevated temperatures in the heat sink area. The temperature is significantly higher towards the fins of the heat sink, an indication that there is a large temperature gradient.

The air flowing at only 2 m/s has too little momentum to transfer heat away from the heat sink efficiently. The blockage of the tunnel of the air causes the increase in heat at the exit (behind the heat sink) to be higher, as well as the warmth discharging at a leisurely pace. A strong thermal boundary layer is then formed as the temperature in the tunnel starts at a large distance from the rear of the heat sink fins. The airflow has not pulled away any thermal energy at all, which implies that the fan does not work very well when it comes to cooling the bulb at low speed. A discernible, gradual decrease in temperature continues from the heat sink to the tunnel exit; however, the clearly pronounced concentrations of heat near the heat sink suggest the lack of a sufficient amount of convective cooling. Due to the flow characteristics, an appreciable percentage of the heat produced by the sink is dissipated inefficiently as the heating element is largely covered with laminar flows of air [25].

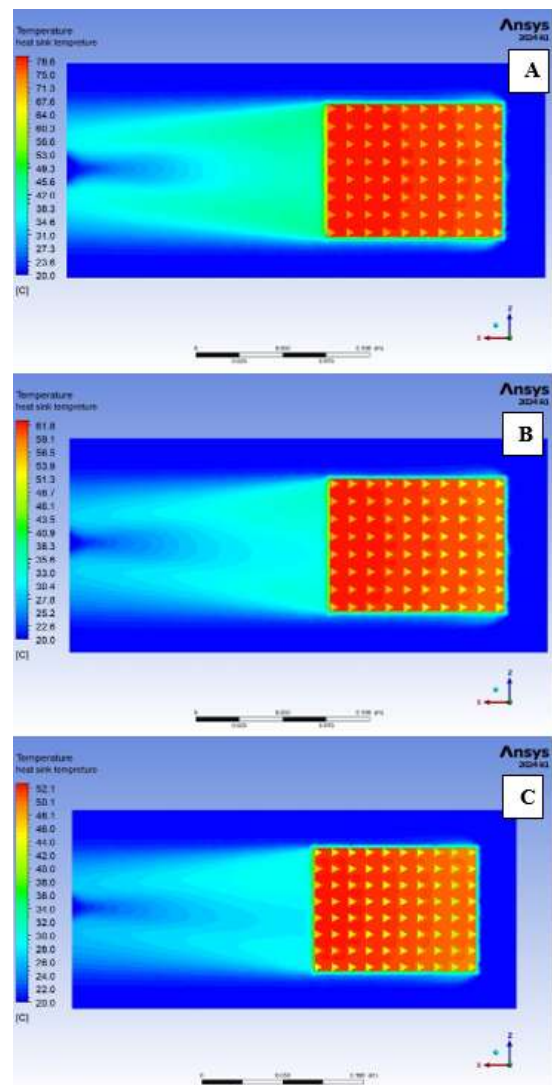


Figure 8. Heat distributions in the heatsink and at the bottom of the air tunnel in cases (A), (B), and (C) with inlet velocities of 2 m/s, 4 m/s, and 6 m/s

In the second configuration with an inlet flow velocity of 4 m/s, the heat removal from the substrate of the heat sink is significantly enhanced by the improvement of extracting of thermal energy extraction and transport into the ambient tunnel. This operating condition minimizes thermal nonuniformity, increases the thermal conductivity, and allows

the flow of air, which is close to the sink, to absorb heat before it is transferred to the downstream regions. Measurements show that the sink surface temperature at 4 m/s is always lower and indicates a better heat dissipation ability from the electronic component. In addition, the reduced boundary layer thickness is also responsible for a further reduction of local air temperature and hence produces a more uniform thermal field behind the sink. The third operating condition, with an inlet airflow rate of 6 m/s, gives the highest heat removal efficiency. Under these circumstances, the zone of elevated temperature is confined to a tiny layer just behind the heat sink so that more thermal energy can be dissipated more rapidly in the surrounding air mass [26]. Thus, the conduit temperature tends to approach ambient temperatures before the exhaust. A steep temperature decrease is seen within the tunnel, with little heat content residue at the sink position.

6.4 Bottom fluid dynamics and thermal transport analysis

The heat exergy transmission from the heat sink is mainly promoted by convective transfer, in which the convective heat transfer coefficient shows an upward trend with the rise of air velocity. At low velocities, a relatively thick boundary layer is formed, which reduces the overall heat transfer rate. In contrast to the suction for the CFD, it is observed that when the free stream is present at a speed of 4 m/s and 6 m/s, the boundary layer thickness decreases, and therefore, the convective heat transfer is increased. Increased turbulent air flow at 6 m/s further increases the heat transfer efficiency due to the increased air flow mixing and corresponding decrease of thermal resistance between the heated sink at the heat sink surfaces. Conductive heat transfer along the heat sink decreases due to a thicker boundary layer, which leads to a steep temperature gradient at 2 m/s. Increasing the velocity results in a high rate of boundary layer disruption and a resultant faster heat dissipation and lower temperature gradient. The fast-flowing air acts like a heat exchanger and removes heat from the heat sink. The highest temperature in the tunnel is found near the heat sink fins, and the lowest temperature is obtained near the tunnel front [27].

6.5 Air tunnel temperature profile analysis

The heat dissipation capability of the heat sink is mainly determined by the convective heat transfer, in which the convective heat transfer coefficient will also increase with the increase of the airflow velocity. At smaller values of velocity, the production of a thicker boundary layer leads to a decrease in the rate of heat transfer. When the velocity of the airflow is 4 m/s and 6 m/s, it will contract the boundary layer and consequently increase the convective heat transfer. Turbulent flow at 6 m/s further promotes heat transfer efficiency because of the better mixing of air and reduced resistance to the flow of air over the heat sink surface. At an air flow speed of 2 appropriately m/s, the higher layer impedes the conduction of heat from the heat sink to create a major gap in temperature. Increased flow velocities cause a thin boundary layer to form rapidly, and heat energy is dissipated more quickly, and a temperature difference. The fast air movement acts well like a heat exchanger, moving the thermal energy away from the heat sink. The highest temperature inside the tunnel is measured near the heat sink fins, while the lowest temperature is near the front part.

6.6 Air tunnel section fluid dynamics and thermal transport analysis

From Figure 9, at 2 m/s, the convective heat transfer rate is inadequate, but it rises significantly with the increase in air velocity to 6 m/s. At the air velocity of 2 m/s, the flow is still laminar in nature, thus forming a thick boundary layer around the heat sink, which decreases the heat transfer coefficient over that air velocity. At an air velocity of 4 m/s, more rapid creation of the boundary layer causes a more rapid attenuation of the boundary layer and the ensuing convective heat transfer, although to a limited extent. As the velocity increases to 6 m/s, the flow becomes turbulent, which vastly increases heat transfer. This leads to a better spread of temperature, so that the air-conditioning system can cool down the living space. Electricity is transferred to the fins of the heat sink, and the thickness of the boundary layer depends on the air velocity. A low-velocity heat transfer coefficient is observed when the resin thickness is low; meanwhile, the heat dissipation effect is enhanced as the velocity increases and the thickness of the boundary layer decreases. Turbulence of 6-m/s increases heat dissipation via breaking the thermal boundary layer, reducing the sink temperature, and enhancing the heat transfer in the air duct. The higher the airspeeds, the faster the thermal diffusion, and so the temperature of the air will be uniform over the duct [28].

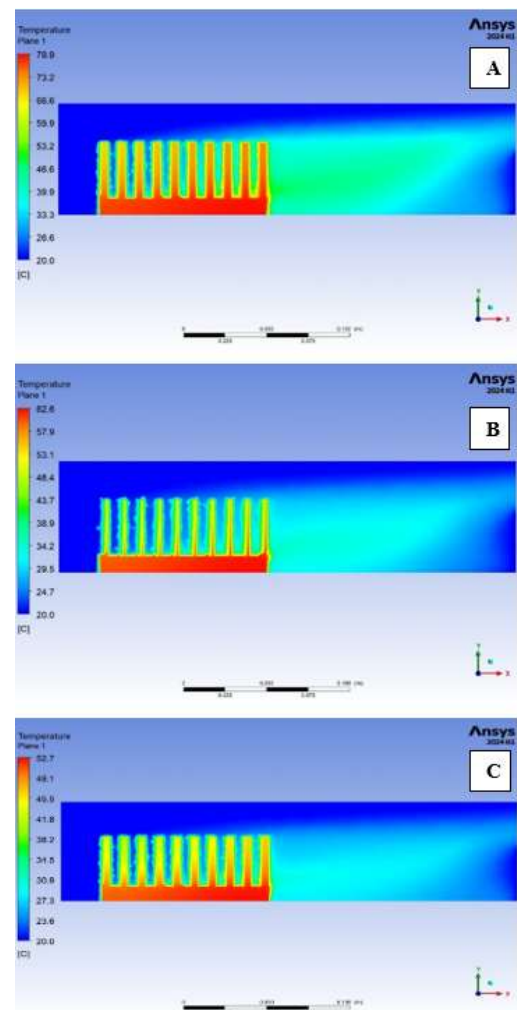


Figure 9. Air tunnel section heat distributions (A) 2 m/s inlet speed, (B) 4 m/s inlet speed, and (C) 6 m/s inlet speed

6.7 Air tunnel outlet temperature analysis

Figures 10(A), (B), and (C) illustrate the temperature variation of the air exiting a heat sink within an air tunnel at three different velocities. Air circulates over the heat sink, and the distribution of heat is governed by the heat flux beneath the heat sink, analogous to a genuine electrical device. The temperature in the tunnel reaches 35.7°C near the heat sink and decreases with increasing distance from it (Figure 10). The transition from red, signifying elevated temperatures, to blue, representing lower temperatures, illustrates the temperature gradient, with the central region adjacent to the heat sink exhibiting higher temperatures and the peripheries displaying cooler temperatures.

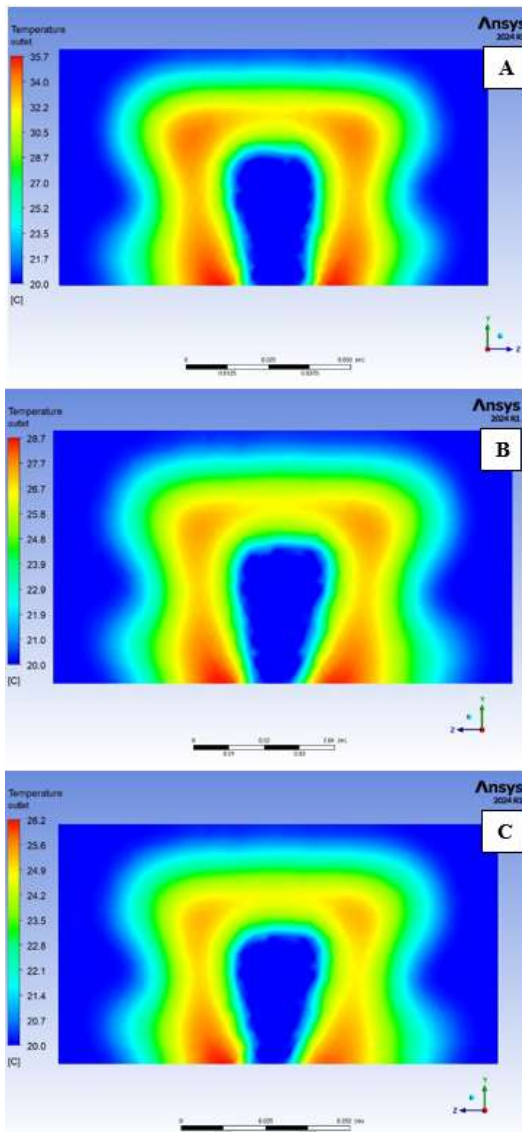


Figure 10. Outlet heat distribution models (A) 2 m/s inlet speed, (B) 4 m/s inlet speed, and (C) 6 m/s inlet speed

The heat sink's temperature gradient is influenced by the air velocity, with a 2 m/s entry air velocity causing a reduced heat transfer rate. The heat sink gradually releases heat, which means a rise in temperature above the surrounding air temperature. However, the air around the laptop is relatively cold due to the resultant low thermal mass interaction. An increase of the airflow to 4 millipascals of air per second (ms^{-1}) leads to an increased convective heat transfer coefficient,

resulting in more homogeneous temperature distributions. As a result, the heat sink emits the heat more efficiently due to the increased fan airflow. At a maximum of 6 m/s, the air temperature in the outward movement is surveyed at 26.2°C . In such conditions, the temperature gradient at the outlet is getting steeper and steeper, and the hot spots next to the heat sink are significantly reduced. Improved performance - the enhanced rotation of the heat sink makes it highly efficient in dissipating the heat, thus enhancing the capacity of the chip to be cooled. This is in keeping with the spatial temperature distribution that confirms adequate thermal transfer when adequate air circulation is present. Figure 10 shows that the outlet temperature decreases with an increase in airflow velocity. With increased axial velocity, it helps in more heat transfer and hence lowers the outlet temperature and, in turn, provides a more uniform temperature distribution. In these simulations, the primary control on the disposition of the temperatures is that of heat transfer from the heat sink to the ambient air. Increased air circulation facilitates the waste heating from the heat sink, thus preventing overheating at the outlet. All of the researches show that the air near the edge of the heat sink is relatively colder than the air at the center, which indicates the reduced heat conduction efficiency at the outer section [29].

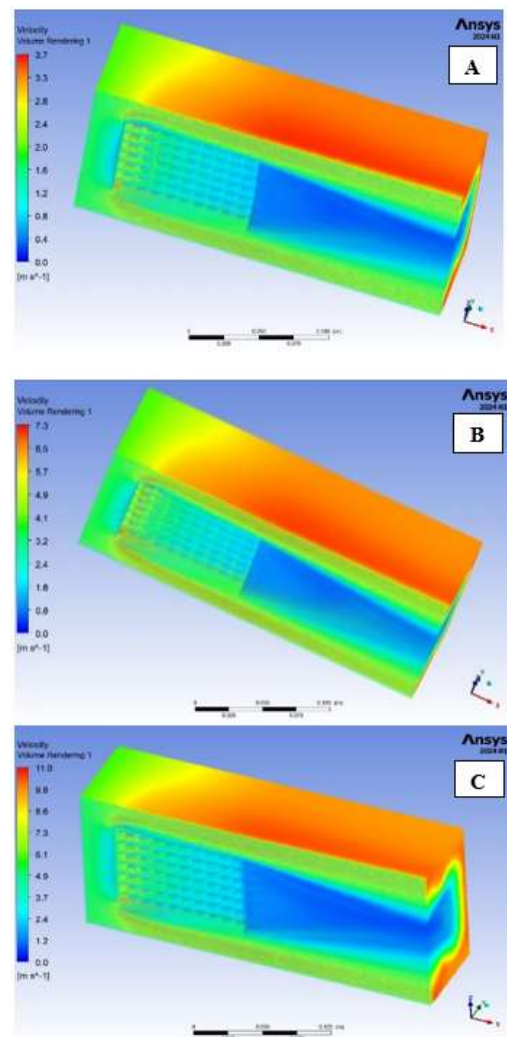


Figure 11. The air tunnel bottom and outer wall velocity distribution is modeled for three inlet velocity conditions: (A) inlet velocity of 2 m/s, (B) inlet velocity of 4 m/s, and (C) inlet velocity of 6 m/s

6.8 Air tunnel Bottom and the outer wall velocity profile analysis

Figure 11 shows how the air speed around heat sink fins reacts when placed inside an air tunnel with air entering at different speeds (2 m/s, 4 m/s, and 6 m/s). This affects thermal management and creates a less turbulent flow around the heat sink. A localized region cannot flow past the surface of the heat sink is created by this limited flow rate, resulting in a reduced convection zone and thus heat transfer. The device also does not dissipate the heat effectively; this may find its result in a hot point adjacent to the heat sink. Moved by a lowered airflow, it allows for less efficient cooling of the heat sink because the flow insufficiently cools its surface, and hence reduces the heat dissipation rate.

The stagnant stream enables the pressure of air to be applied on all sides of the heat sink, which makes the temperature in the area high. In the 4 m/s scenario (Figure 10(B)), the air speed is more balanced as opposed to the 2 m/s scenario. The heat sink liquid has a faster velocity and increased movement around the fins, especially in the downstream. The flow velocity of 4 m/s causes random movement of the components of the media, which promotes mixing and transfer of heat. A higher speed of the fan allows it to take out more heat in the heat sink and release it to the outside. The circulation of air around the heat sink is also increased, which is observed through the high gradient of the velocity field. When there is more airflow, the heat sink is viewed to have a much more efficient cooling ability. Increased velocity of air above the fins maximizes its thermal transfer capacity, thereby increasing thermal dissipation. The optimum functioning of the heat sink is when the air velocity is as high as possible, and in this manner, the flow velocity is not skewed in any part of the heat sink. Such an even distribution enables the highest velocities to touch the whole surface. The high air flow around the heat sink is conducive to the mixing and cooling of the heat. Turbulent flow reduces hot spots on the surface of the heat sink, which can cause possible damage to the components. The maximum cooling of the heat sink is attained by the maximum heat dissipation, which does not allow stagnation in the parts, which might cause a bottleneck [30].

6.9 Air tunnel section velocity profile analysis

The findings of ANSYS modeling of the air flow around a triangular fin-shaped heat sink in an air tunnel at different inlet air speeds (2 m/s, 4m/s, and 6 m/s) are shown in Figure 12. These pictures represent the attributes of airflow around the heat sink and the effect on the rate of heat dissipation out of the heat sink. Figure 12(A) shows blue colors predominating around the heat sink and the bottom of the image in case the inlet velocity is 2 m/s, and the surrounding airflow is stagnant. At the top of the tunnel (red), the air flow is fast, and the airflow does not go through the heat sink. The lower velocity near the heat sink (0.8 to 2.4 m/s) indicates the presence of a very stagnant fluid on the edges of the heat sink. These places have a lot of air stagnation that prevents them from cooling down the surrounding environment in an effective way. This, in turn, raises the temperature of the heat sink, thereby reducing the effectiveness of the cooling role. The slow velocity of air leads to the failure of transmission of heat efficiently between the heat sink and the air. The laminar flow lowers the amount of heat that is transmitted and hence increases resistance to heat. This causes less heat to dissipate, which may overheat the device, hence overheat the heat sink.

On green and yellow patches that surround the heat sink at an air speed of 4 m/s, the deeper the airflow, as high as 5.5 m/s, is useful in heat dissipation. Turbulence generated over the heat sink promotes efficient heat extraction from the CPU. Triangular fins create a swirling vortex, optimizing surface heat transfer. An enhanced airflow design improves the cooling efficiency of the heat sink, enhancing its capacity to transfer heat to the air and remove it more efficiently. High air velocity enhances greater air mixing, leading to more homogeneous surface temperature and also lesser thermal resistance. This airspeed will cause rapid cooling and hence increased mixing and faster heat transfer, reducing stagnant air and causing uniform heating in the entire tunnel.

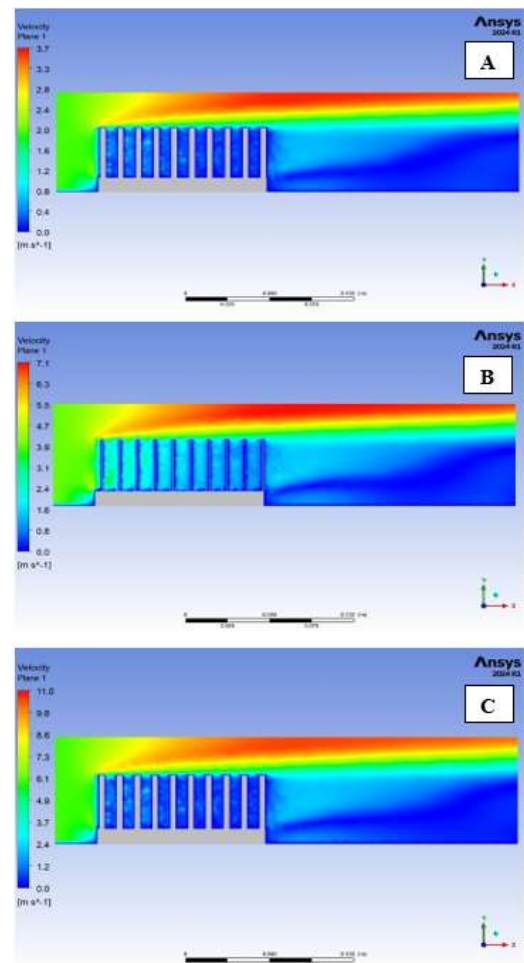


Figure 12. Model of air tunnel section velocity distribution (A) 2 m/s inlet speed, (B) 4 m/s inlet speed, (C) 6 m/s inlet speed

6.10 Outlet velocity profile analysis in air tunnel

Figure 13 shows the distribution of velocity at the tunnel outlet with three triangular-shaped fin heat sinks in place, and an electrically powered apparatus at the bottom releases heat to the vicinity air. Spatial variation of velocity in the duct is investigated in three different inlet air velocities: 2 m/s, 4 m/s, and 6 m/s. The flow field in Figure 13(A) builds up with the length of the tunnel. In the central part of the image, all of the velocities are between 0.4 and 2.4 m/s, and the color scheme suggests that the tunnel walls and edge areas have the greatest velocities and reach 3.7 m/s. Some of the air near the heat sink produces significantly slower speeds. The triangular inserts bring about a localized turbulence, though at this low flow rate, the effects on the overall performance are also minimal.

The slow-moving air results in inadequate circulation in the heat sink, hence hindering the quick removal of surface heat. This means that the heat sink is driven in a mostly laminar regime; hence, it reduces mixing of the surrounding air and reduces its ability to dissipate heat. The conditions lead to intense thermal sources on the surface of the sink, which makes it difficult to cool the equipment.

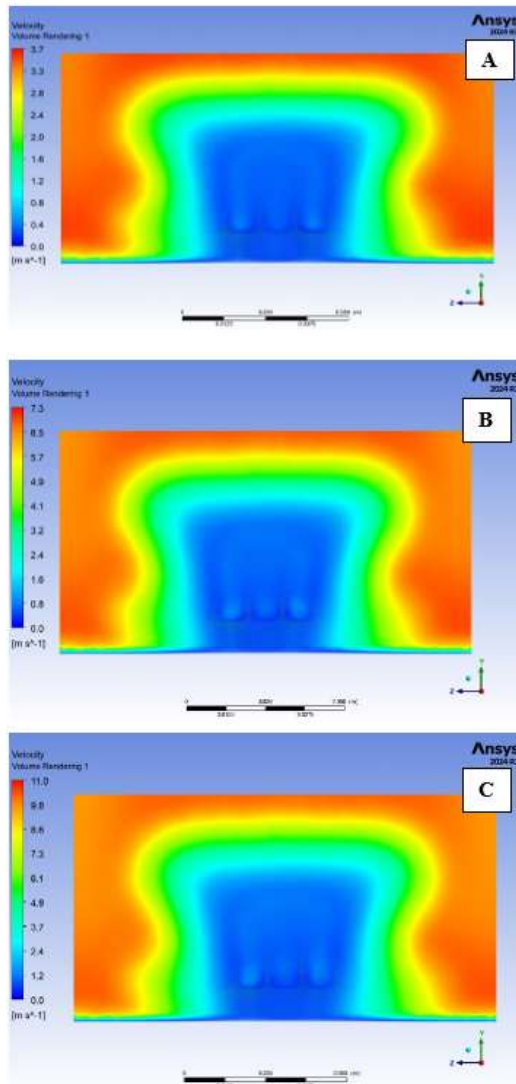


Figure 13. Air tunnel outlet velocity distribution models (A) 2 m/s inlet speed, (B) 4 m/s inlet speed, and (C) 6 m/s inlet speed

Figure 13(B) shows a much higher flow rate as compared to Figure 13(A). Velocities are between 2.4 and 4.9, when it was 3–5 m/s in the area around the heat sink. The field of velocity around the heat sink is smoother, and the number of fins causes significant turbulence, and in a result increase in the flow of the surrounding air. With an inlet velocity of 4 m/s, turbulence induced by the outflow velocity raises the stagnated sheet of air near the heat sink and enhances heat exchange. The improved passage of air, in turn, improves the dissipation of the heat out of the system, results in a more balanced temperature distribution, and alleviates the presence of hot spots. This has the overall effect of minimizing the temperature of the component attached to the heat sink. Figure 13(C) indicates the highest velocity of 11 m/s, the highest velocity of flow that was attained during this research. Within the center portion of the heat sink, it will create a uniform velocity distribution through the domain as a result of the

vigorous mixing. Heavy turbulence and deep mixing are also witnessed in the fluid at an inlet velocity of 6 m/s, which enhances convective heat transfer even better.

The air velocity is rapid to facilitate the swift dissipation of heat from the heat sink. Enhanced turbulence compels the heat boundary layer to alter continuously, resulting in more consistent heat transfers and improved performance. Due to its rapidity, the engine dissipates heat swiftly and effectively. The heat sink performs optimally in cooling due to the rapid airflow that dissipates heat from its surface. As a result, the heating of the heat sink is more homogeneous, which will guarantee the electronic equipment to be in a safe thermal temperature range. Convective heat transfer is improved as the rate of input air flow is enhanced, hence increasing the efficiency of the heat sink to dissipate heat to the chip. Turbulence is increased, thus enhancing the mixing of air, hence making the heat sink exchange a larger amount of heat with the air around it. When the translation velocities are low, the flow field has lower uniformity; conversely, as the velocity rises, the flow field gets more uniform [31]. With low speeds (around 2 m/s), the airflow stagnation will follow, which will cause the heat to accumulate inside the area of the heat sink. When the air currents accelerate to 4 m/s and 6 m/s, the flow becomes increasingly turbulent, facilitating enhanced heat distribution. Optimal cooling is achieved when the fan operates at maximum velocity (6 m/s), since this generates increased turbulence and enhanced air circulation. It ensures that the heat sink dissipates heat rapidly, preventing excessive thermal accumulation near the heat source.

6.11 3-way air inlet temperature profile analyzes

Figure 14 shows a temperature distribution in a heat sink because of a 10,000-watt heat source at the bottom. The air tunnel with a heat sink inside has an air inlet velocity of 2 m/s from the front and the two sides. This design shows that a thorough study of the heatsink thermal performance under forced cooling should be carried out. The color contour scale shows the heat sink has a temperature range between 38.5°C to 57.0°C, and the zones show the colder temperatures in blue and the hotter temperatures in red. Through this contour, authors can study how the heat generated by the base is transferred through the fins to be distributed by the air around the heat sink.

Convective heat transfer in a system depends on the speed of air entering the system. At 2 m/s, forced convection transfers heat faster due to uniform air movement. This design helps remove more heat by creating airflow in various directions. Faster airflow speeds increase the thermal boundary layer thickness at the heat sink, speeding up heat transfer. Heat across the fins is uneven, with temperatures rising from the base and decreasing as air rises. The area near the base has the steepest temperature gradient. The center of the heat sink is generally hotter than the edges due to turbulent and slower-moving air. Figure 14(B) shows the temperature distribution in a system, with hot regions in red and cool areas in blue. The hottest region is at 56.5°C, while the coldest is at 26.8°C. The heat sink fins have the highest temperatures, with the hottest being at the core. The air in the frontal flow zone is chilly, indicating deficient thermal energy. The gradient shows temperature variation as air is transported from the entrance to the heat source. Air enters the heat sink to provide uniform velocity and temperature, reducing the likelihood of local hotspots. Turbulence and boundary layer disruption are crucial for achieving a high local Nusselt number. Heat transfer

occurs more rapidly on the back fins, providing lateral circulation for cooling. Heat extraction improves with increased turbulence and Reynolds number in an aircraft featuring compound inlets [32]. The fins exhibit stability and consistent heat dissipation outward. The light blue hue of the wake indicates significant convection heating.

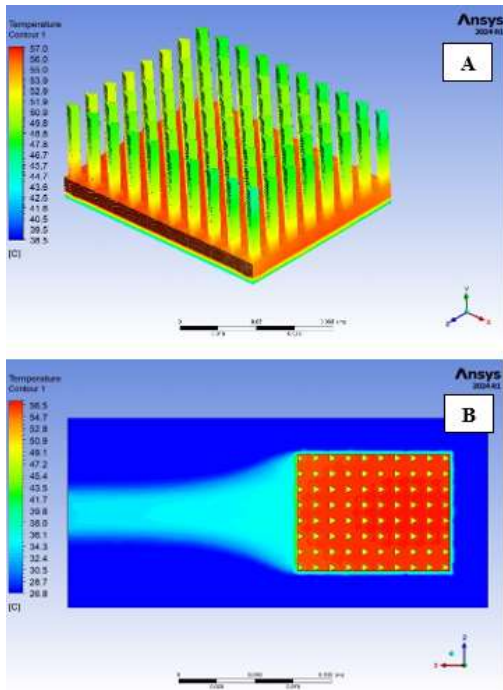


Figure 14. (A) 3-way 2 m/s air inlet heat sink temperature distribution profile, (B) 3-way air inlet heatsink and air tunnel bottom heat distributions

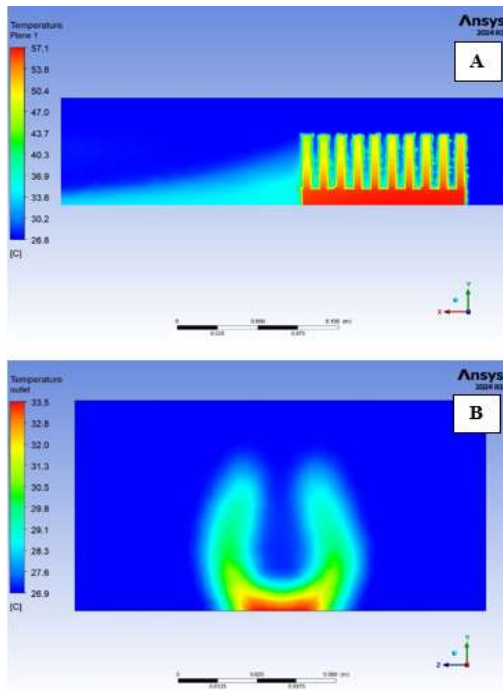


Figure 15. (A) 3-way air inlet heat sink and air tunnel cross-section heat distribution, (B) 3-way air inlet, air tunnel, and outlet temperature distribution

The XY cross-sectional view of the heat sink, as shown in Figure 15(A), illustrates the presence of fins inside the tunnel,

so this is used in the analysis. The electronic device absorbs heat using vertical fins, which promotes the heat flow to the surrounding air, hence cooling the electronic device. The strengthened thermal performance of the heat sink is associated with forced convection. There are three different directions in which ambient air is introduced, and this guarantees equal distribution of heat as well as effective dissipation. The tri-inlet configuration spreads the thermal load and hence prevents the overheating risk, which in turn preserves the performance of the system. The inflow of hot air at the upper, lateral, and inferior positions disturbs the thermal layer in the exterior of the fins, causing inefficient heat transfer. The intake structure has a checkboard form, which reduces the thickness of the boundary layer to a minimum, increasing heat loss through the fins. The color gradient of the fins is red (at the base), blue (at the end), indicating a progressive fall in temperature as one goes closer to the base than the apex. Figure 15(B) shows the temperature distribution at the heat sink outlet assembly ranged between 26.9°C (blue) and 33.5°C (red). The experimental data shows that the heat sink dissipates the heat to the surrounding air successfully. The high temperature record in the area surrounding the heat sink, where the thermal energy is concentrated. When the air reaches the heat sink, the temperature of the air will reduce as it gets farther, explaining that the outgoing air is much cooler in comparison with its original temperature. The spatial pattern of the temperature of the air in the tunnel provides an assessment criterion of cooling performance and thermal structuring of the system.

The peak temperature would be expected to be close to the heat sink, which is how heat is transferred through the process of exchanging it between the source and the environment [33]. The cooling system uses forced convection, and the air flow enters the system via three discrete ports at a speed of 2 m/s. Increasing the speed of air passing over the fin enhances mixing, decreases the layer of stagnant hot air close to the fin blades, and improves the efficient and quick removal of heat from the heat sink.

6.12 Velocity profile of a three-way air inlet

The results of the CFD simulation of air leaving a heat sink that has triangular-shaped fins indicate a clear and uniform water velocity within the intake area. The air velocity ranges between 2.1 m/s to 18.5 m/s at the intake, which evidences shifting slowly moving air, rapid shifting to the limit, and swiftly moving air. When the air touches the heat sink fins, the speed of the air changes and hence causes turbulence to the fluid in the surroundings. Regions just out of the fins where streamlines are constricted are the ones with increased velocities compared to other sections. The sharp edges are classified as rapid acceleration because of an increased velocity in the particular area. The water then moves around the triangular-shaped fins to increase its speed, which reaches a maximum of 18.5 or so towards the fins. The fins offer a narrowed channel to the flow and thus increase the velocity of the flow in that area and consequently cause high velocities. The speed of air decreases towards the fin, thus resulting in a slowing of the flow and high turbulence. Massive drag in velocity is experienced at the surfaces of the fins due to the effect of the boundary layer. As the distance of the fins increases, air pressure and velocity increase once more. The high to the low velocity shifts are sudden, with a velocity of 18.5 m/s at the tip of the fin and either 10.3 m/s or below in

the wake.

The thermal radiations given by the base of a heat sink are used to increase the temperature of the surrounding air, which causes the development of a thermal boundary layer. This layer makes the air near the heat sink flow at a lower speed; thus, a reduction in the speed of the surface wind occurs. The ambient airflow gets disrupted by the upward movement of the hot air in the heat sink, creating the fluctuation of air velocity in a stipulated area. Streamlines can separate as the convection moves the hot air to the top, and velocities of about 10 m/s and some more may develop, as the hot air is accelerated because it is of lesser density. The thermal flow zone can also cause mixing between the heated air and the surrounding air, and this would cause an alteration in the velocity pattern. These thermally induced currents can form certain areas where there is an augmented current in comparison to the nearby cooler spaces. The air velocity is enhanced by a few millimeters per second due to the convection process, whereby it increased to 12.3 m/s instead of 6.2 m/s. Towards the periphery of the heat sink, the walls and triangular fin structure that surrounds the construction affect the airflow. Inside the tunnel, the air is moving at high velocities, between 10.30 m s^{-1} and 18.50 m s^{-1} , as fewer objects may occur. Conversely, the air near the sidewalls and fins has a lower velocity as compared to the remainder of the flow [34].

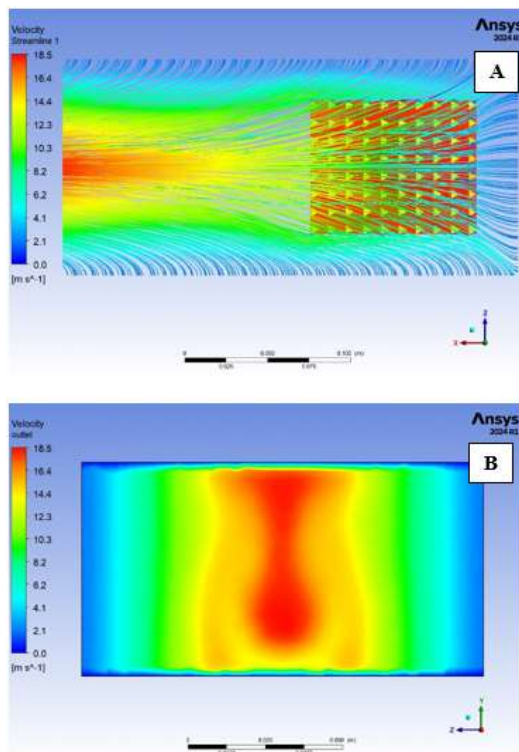


Figure 16. (A) 3-way air inlet velocity profile. (B) 3-way air inlet tunnel outlet velocity profile

Figure 16(B) exemplifies the velocity distribution (at the exit of the air tunnel) of an air stream through the heated fins and the variation in the distribution of the velocity. The shape of the heat sink, as well as the contact with the fins, influences the flow out of the outlet. During the intermediate part, the velocity of the air mass is extremely high (18.5 m/s). However, the funneling effect of outflow suggests a constricting point, or a neck, in the middle of it, which suggests that the flow has hastened as it leaves the heated area. The velocity decreases gradually beside the tunnel walls, with a minimum of 2.1 m/s.

The heat flux conducted through its lower surface changes the velocity across the heat sink. Through the heat gradient, a very thin layer of air builds up, creating quicker air flow. The flow rate of 18.5 m/s by the inner stream means that the flow is accelerating when it flows around the sink. Heat sink and triangular fins probably increase the level of operation of fans due to the reduction of the aperture.

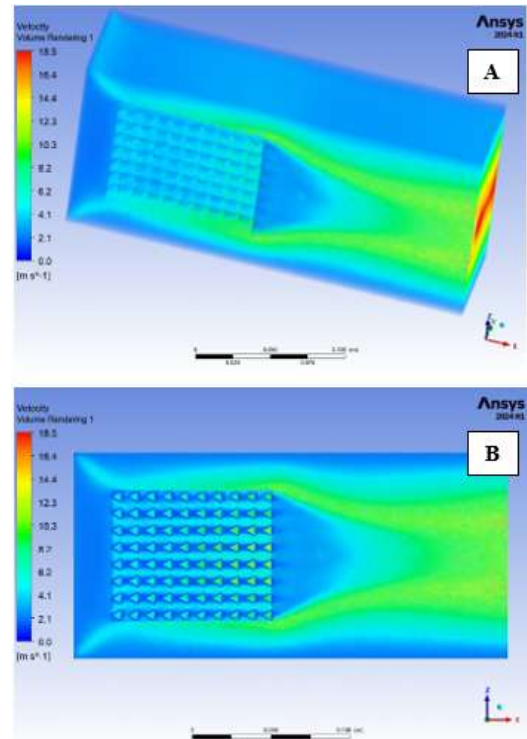


Figure 17. (A) and (B) bottom, walls, and outlet 3D 3-way air inlet tunnel velocity model

Figure 17 shows the air flow in the air tunnel by giving a comparative analysis of the velocity of air and the working mechanism of the mechanism of cooling system, that is, the heat sink and the triangular fins. The flow of air moves through the tunnel entrance, causing a flow, and it reaches the heat sink, coming into contact with the fins, which causes strong differences in the velocity. The sharp edges of the triangular fins create high velocity streams of air, and turbulence caused by the fins increases the speed of central airflow, but at the same time slows it down in the deep downstream areas.

Having moved through the triangular fins, the flow displays acceleration, resulting in regions of significantly lower speed. These zones are reflected as spectral coloring, with the blue color representing the areas where the airflow is less, as the air is no longer attached to the fin surfaces. The thermal layer is formed on the surface of the heat sink, which is due to the release of heat at the bottom of the heat sink, and this cools the air, causing a slowing of the air temperature. The resultant loss of velocity is within the range of 4.1 m/s to 6.2 m/s. Thermal effects are also noticeable within the vicinity of the source of heat, and in places where the air rises.

After the airflow arrives on the tunnel walls, it is further slowed down by viscous friction in the boundary layer, and is sufficient to reach such wall velocities as 2.1 m/s: this is due to the confinement effects imposed by the tube shape. The flow along the walls is slower than at the middle of the tunnel. Triangular fins accelerate airflow, constricting the airflow and enhancing velocity in their vicinity. Turbulence generated by

fins creates vortices downstream, establishing areas of diminished velocity. The streamlining is ineffective due to the presence of turbulent eddies behind each fin, impeding flow, and reduced velocity. Wake regions contribute to enhanced mixing and heat transmission throughout the system. Heat is delivered near the base of the heat sink, resulting in reduced air density.

6.13 Model assumptions affect

Some assumptions and simplifications are made on the basis of the research model that can affect the accuracy and relevance of the results. The heat sink and fins were designed using a simplified geometry of idealized triangles, fin shapes in SOLIDWORKS, although real-world manufacturing failures and highly complex real-world surface finishes. The simplification can cause a bias in the results when analyzing a uniform geometry of a fin, i.e., underestimation of effective heat dissipation. The thermophysical properties, thermal conductivity, and emissivity, are likely to remain stable and temperature independent and might affect the predictions at different operational temperatures. The air was regarded as an ideal gas that is incompressible and whose flow is in steady-state turbulent flow, without taking into account the conditions of transience and little compressibility that might arise in short-term changes in heat. It was simulated in the realizable k - ϵ model, which is deemed a decent compromise between the accuracy and the cost of computation. This model, however, averages small-scale eddies and can also fail to involve small-scale turbulent structures and near-wall effects relative to more complex models, such as LES. The steady-state approach overlooks any transient thermal change and temporary heat transfer change in the transient state of real electronics cooling processes, when temporary thermal change and dynamic airflow change are commonly experienced in practice (usually in electronics cooling). In the case of the computational domain, some spatial limitations were imposed that include the size of air tunnels and no side gap between the heat sink base and the tunnel base, which cannot adequately characterize all actual installation environments. The simplification of these boundary conditions may have an impact on the behavior of airflow and heat transfer, which is used to predict the heat sink performance. Overall, despite consideration of the modeling and geometry assumptions, which give a good understanding of the mechanisms of heat transfer to illustrate the advantages of turbulent airflow over triangular fins, the idealization and simplification of both models can change the outcomes, which can influence them, and experimental validation under realistic conditions is suggested to check the results.

7. RESULTS COMPARISON

As a result of the simulations, the researchers compared the outcome of the 6 m/s velocity enhanced by a single air inlet to that of the 3-way air inlet systems. The analysis will look at two major aspects: temperature and airflow levels in every area, especially where the triangular fins are located in the heat sink.

7.1 Heat distribution comparison

At a velocity of 6 m/s, the heat sink exhibits a limited temperature range of around 41.6°C to 52.5°C. The enhanced airflow facilitates expedited and more efficient heat transfer.

As turbulence intensifies and the boundary layer diminishes at 6 m/s, the heat sink can dissipate heat more rapidly. The temperature concentration all over the heat sink is smoother, especially in areas that are far away from the base in where heat is built up when the operating speed is low. Setting the fan speed to 6 m/s produces significant turbulence, hence enhancing heat dissipation. Consequently, the heat sink's surface temperature decreases, resulting in reduced temperature gradient across it. Increased turbulence diminishes the likelihood of hot spots emerging, as they often occur at lower airspeeds. When the model 3-way intake operates at an airspeed of 2 m/s, the heat sink temperature varies between 38.5°C and 57.0°C. In this scenario, less heat transmission occurs due to the slower arriving air, which creates a thicker insulating barrier surrounding the items. The inclusion of three air ducts from the front and sides in the heat sink enhances the uniformity of temperature distribution over the whole heat sink. The accelerated increase in air temperatures and the more uniform heating are attributed to the multi-directional airflow. Temperatures at the base have been rather stable; yet, air at all elevations now retains warmth for an extended duration. At this velocity, the air circulates at a reduced rate, hence diminishing the efficacy of forced convection. Nonetheless, the numerous inlets facilitate air circulation in various directions; nonetheless, this does not yield as uniform heat dispersion as a velocity setting of 6 m/s.

7.2 Velocity distribution comparison

6 m/s inlet model shows 18.5 m/s velocity at the middle of the air tunnel, significantly due to rapid acceleration past the heat sink. The rapid airflow causes significant turbulence in the liquid within the section, and the streamlines around the fins are constricted, indicating that the velocities around the sharp points of the fins are maximized. Adjacent to the tunnel walls, the velocity decreases to 2.1 m/s, resulting in a more pronounced boundary layer effect. Scientists predict this condition in places where airflow is near the surface in constricted geometries since the airflow velocity is reduced in such places. The fast flow of air causes turbulence and therefore increases mixing and transfer of heat. The fins cause turbulations in the air near the surface, which results in more effective heat dissipation of the entire portion of the component. The speed of the 3-way inlets is also lower than the single inlet, especially at 2 m/s; however, the air flow through the heat sink moves smoothly. The velocity of airflow can reach 18.5 m/s within the core, but it decreases near the fin tips and the walls of a container. When the air flows along the sharp edge of the fins, the fins enhance high movement. The fans create the agitation that results in making the air near the radiator fins reach up to 12.3 m/s in a single direction. On the other hand, the flow in the back of the fins reduces to 6.2 to 2.1 m/s to produce an impact on thermal dissipation efficiency of the system.

The shape of the fins at the various inlets helps in better mixing of hot air around the heat sink. The unique air inlets in the three-way model encourage evenly distributed airflow within the cavity as compared to the single inlet design of the cavity, with the airflow predominantly taking a single path. Table 4 provides a comparison between airflow and heat dissipation values based on the 6 m/s single inlet design and the 3-way inlet model that is running at a speed of 2.9 m/s. It determines important parameters such as the pattern heat, the dissipation rate, and the effect of triangular fins on the velocity of a flow.

Table 4. Velocity profile and heat dissipation of the inlet airflow velocity at 6 m/s and three-way air entry with 2 m/s air entry velocity comparison

Parameter	6 m/s Single Inlet	3-Way Inlet (2 m/s)
Heat Distribution - Temperature Range	41.6°C to 52.5°C	38.5°C to 57.0°C
Heat Dissipation Efficiency	High - Efficient heat dissipation, especially from the base and fins	Moderate - Less efficient compared to a 6 m/s single inlet
Air Velocity at the Heat Sink	Up to 18.5 m/s, high acceleration near fins	Up to 18.5 m/s in core areas, but lower overall due to lower airflow speed
Air Velocity at Tunnel Walls	Down to 2.1 m/s	Down to 2.1 m/s, but more uniform due to multiple inlets
Turbulence and Mixing	High turbulence, enhanced mixing, and heat transfer	More uniform distribution, but less turbulence, less effective mixing
Temperature Uniformity	More uniform temperature distribution, especially at the fins	More uniform temperature distribution, but still hot spots near the base
Effect of Multi-Inlet Design	A single inlet creates a concentrated flow with high velocity	Enhanced uniformity due to air entering from three directions
Effect of Triangular Fins on Velocity	Fins induce local accelerations up to 18.5 m/s, and turbulent flow helps heat dissipation.	Air reaches 12.3 m/s near the fins, but the wake regions experience slower air (down to 6.2 m/s)

Table 5. Triangular fins result in comparisons with the previous studies

Fin Shape	Key Findings	Relative Performance
Rectangular	Simple to manufacture and high overall heat transfer rate, but prone to hot spots and heavier in mass [4, 35].	Efficient at moderate flow but less uniform and prone to local overheating.
Circular (Pin Fin)	Efficient airflow with less resistance, works well at moderate-to-high Re, but has a lower surface area and weaker turbulent mixing [36].	Good at removing heat at high flow, but limited surface interaction for thermal exchange.
Triangular	Higher surface area per mass, enhanced turbulence and mixing, uniform temperature, lightweight; performs best under turbulent flow [37].	Superior at high-speed turbulent airflows, achieving the lowest peak temperatures.

It is possible to compare the different performance of thermal performances relating to the airflow and intake configurations described. This kind of analysis can explain the effect that the airspeed and the type of inlet might have on cooling efficiency.

8. VALIDATION OF ANSYS SIMULATION RESULTS

The data in the study on the triangular heat sink fins were assessed by the authors against the previous works of other researchers in order to confirm their relevance. The simulation by the use of computational fluid dynamics proves performance and fluid dynamics in the heat sink.

8.1 Effect of triangular fins on heat transfer performance

Based on numerical calculations, triangular fins are greatly beneficial to increase heat sink capacity and dissipation of thermal energy by modifying the boundary layer and causing greater turbulence. The investigation by Menni et al. [35] studied the effect of fin geometric features, such as a triangular fin, on turbulent heat transfer with respect to fin spacing. The authors argue that triangular fins offer more surface area and geometric complexity, thus enhancing heat-transfer efficiency at turbulent flow conditions [35]. The simulations below show that a simulation in which the airflow rate is increased will increase the rate of heat transfer to the surroundings of the device faster and will thus support the central role of the unique geometry of the fins in improving cooling effectiveness. The triangular shape performs better than circular and rectangular ones due to the enhanced turbulence generated by the shape that facilitates mixing of air, destabilization of thermal boundary layer and provides more

uniform temperature conditions on the surface of the heat-sink. The results of the CFD presented in this paper indicate that as the air velocity decreases, it rises to 2 m/s to 6 m/s, and the temperature of the hottest part of the heat sink is reduced to values as low as 52.5°C. This is credited to the fact that triangular fins form a smaller boundary layer, and localized air acceleration (later on 18.5 m/s in and around the tips of the fins) increases the intensity of the convective heat transfer. These increased speeds ensure that the flow develops into full turbulence, and the triangular geometry facilitates the formation of vortexes and recirculation of the air that facilitates heat dispersion. Table 5 compares the results with the previous research papers' results for more accuracy of these results.

8.2 Comparison with heat sink performance

The study's finding is corroborated by Kishore's research on the effectiveness of triangular fins in CPU heat sinks. Kishore's computer simulations demonstrated that triangular fins disperse heat far more efficiently than rectangular fins, particularly under varying circumstances or airflow. The results of the simulation in this study indicate that triangular fins enhance convective heat transfer, particularly at elevated air velocities [36].

8.3 Validation of flow and thermal characteristics

These fluid dynamics and heat transport in the proximity of the triangular fins are in line with the results of Hithaish et al. [37]. Their numerical study encompassed CFD simulations of triangle pin fin heat sinks, where it is examined that triangle fin types support greater fluid mixing and additionally decrease the thickness of the boundary layer of heat, thereby

promoting greater heat transfer. The current results are consistent with the results reported by Hithaish et al. [37], which highlight that dissipation of heat is easier due to the higher rate of airflow.

8.4 Improvement in cooling efficiency

Hameed et al. [38] investigated the efficacy of heat sinks, including diverse triangular fin configurations, utilizing computational fluid dynamics (CFD). The study indicates that triangular fins facilitated airflow that is more equitable and enhanced cooling efficiency. The study indicates that triangular fins provide more uniform heating throughout the heat sink, particularly under conditions of increased airflow. In summary, the simulation statistics corroborate prior research demonstrating that triangular fins enhance the performance of heat sinks. Menni et al. [35] asserted that triangular fins enhance cooling by increasing surface area and inducing more turbulence. Bhuvan and Kishore [36] concur with the findings, indicating that triangular fins are more effective at cooling than rectangular fins. Hithaish et al. [37] and Hameed et al. [38] showed via their experiments that triangular fin heat sinks enhance heat dissipation by optimizing fluid flow, consistent with this research.

9. CONCLUSIONS

The simulation helps to provide important information on the role of turbulent air velocity on the exit of a heat sink in its cooling efficiency. A rise in the inlet air velocity from 2 m/s to 6 m/s significantly lowers the temperatures in the heat sink, therefore enhancing thermal control. The study highlights the fact that triangular fins help the airflow, break the stable layer near the heat sink, and contribute to the smooth distribution of temperatures all over the heat sink.

A. The research results indicate that heat dissipation has important implications on the rate of air-inflow in a computer, and the temperature at the bottom of a heat sink reduces with increasing velocity between 2 m/s and 6 m/s.

B. Triangular shapes promote and increase the rate of heat dissipation by promoting more turbulence, breaking up the boundary layer, and promoting mixing of air in the duct. In such a method, there is no hotspot, and the heat sink is cooled.

C. The faster the air velocity got 6 m/s, the more effective the turbulence became in enhancing heat transfer in the convection. High air speed cooling was found to be more effectively achieved through triangular fins because they could lessen the flow of heat in the hold on to electronic equipment.

D. The analysis examined the heat sink's performance with one stream of air at 6 m/s and a 3-stream setup at 2 m/s. A single inlet provided a smoother heat pattern and better heat removal, while a 3-way inlet configuration provided lower velocities but less effective mixing.

The triangular shape of the fins made the air travel faster, leading to greater turbulence. This turbulence of the air flow played a major role in the dissipation of heat within the heat sink. The single inlet system was at 6 m/s, which enabled a higher velocity (18.5 m/s), which caused dissipation of heat to be more effective.

10. SUGGESTION FOR FUTURE WORK

Material variation: Even better could be the testing of

alternative materials with a higher thermal conductivity than aluminum, e.g., copper or advanced composites, to increase the amount of heat dissipated and minimize the thermal resistance.

Spacing and arrangement fin: Experimentally investigate the influence of various spacing of fins, thickness, and arrangement on the airflow structures and thermal performance of flipping occurrence. Effects of optimization of these parameters can lead to better turbulence generation and reduced pressure drop.

Techniques of manufacturing: The investigation of new fabrication techniques that would allow the creation of complicated triangular fin geometries at smaller scales or with openings to achieve the most surface area with minimal increase in weight or cost.

Transient and multi-physics studies: The simulations can be extended to the transient condition, radiation effects can be considered more extensively, and the thermal analysis of electronic components can be coupled to assess the real operating conditions.

Stage design: Research into the most effective ways of utilizing the benefits of multiple inlets or other passive means of flow control, like synthetic jets or vortex generators, to achieve the creation of increased turbulence and the generation of cooler surfaces in uniformly distributed locations within the confined spaces.

Machine learning optimization: With the use of optimization algorithms as well as the application of machine learning algorithms, one will be able to replace the traditional trial-and-error approaches, and it will be possible to identify the optimal fin geometries and airflow configurations precisely.

REFERENCES

- [1] Soloveva, O.V., Solovev, S.A., Shakurova, R.Z. (2024). Numerical study of the thermal and hydraulic characteristics of plate-fin heat sinks. *Processes*, 12(4): 744. <https://doi.org/10.3390/pr12040744>
- [2] Khor, C.Y., Rosli, M.U., Nawli, M.A.M., Kee, W.C., Ramdan, D. (2021). Influence of inlet velocity and heat flux on the thermal characteristic of various heat sink designs using CFD analysis. *Journal of Physics: Conference Series*, 2051(1): 012013. <https://doi.org/10.1088/1742-6596/2051/1/012013>
- [3] Ranjan, A., Gupta, V., Bagri, S. (2017). CFD analysis of perforated heat sink fins. *International Journal for Research Publication & Seminar*, 8(1). <https://www.researchgate.net/publication/330824008>
- [4] Mirapalli, S., Kishore, P.S. (2015). Heat transfer analysis on a triangular fin. *International Journal of Engineering Trends and Technology*, 19(5): 279-284.
- [5] Arslan, S., Pulat, E. (2023). Computational investigation of the effect of different speed and heat flux parameters on the cooling of the electronic circuit board. *International Science and Art Research Center*, 5: 147-159.
- [6] Huang, L., Yeom, T., Simon, T., Cui, T. (2021). An experimental and numerical study on heat transfer enhancement of a heat sink fin by synthetic jet impingement. *Heat Mass Transfer*, 57: 583-593. <https://doi.org/10.1007/s00231-020-02974-y>
- [7] Hemptijid, T., Kittichaikarn, C. (2020). Effect of heat sink

- inlet and outlet flow direction on heat transfer performance. *Applied Thermal Engineering*, 164: 114375.
<https://doi.org/10.1016/j.applthermaleng.2019.114375>
- [8] Adhikari, R.C., Wood, D.H., Pahlevani, M. (2020). An experimental and numerical study of forced convection heat transfer from rectangular fins at low Reynolds numbers. *International Journal of Heat and Mass Transfer*, 163: 120418.
<https://doi.org/10.1016/j.ijheatmasstransfer.2020.120418>
- [9] Qian, Z.Q., Wang, Q., Cheng, J.L., Deng, J. (2018). Simulation investigation on inlet velocity profile and configuration parameters of louver fin. *Applied Thermal Engineering*, 138: 173-182.
<https://doi.org/10.1016/j.applthermaleng.2018.02.009>
- [10] Pourdel, H., Hassanzadeh Afrouzi, H., Akbari, O.A., Miansari, M., Toghraie, D., Marzban, A., Koveiti, A. (2019). Numerical investigation of turbulent flow and heat transfer in flat tube: Effect of dimples with operational goals. *Journal of Thermal Analysis and Calorimetry*, 135: 3471-3483.
<https://doi.org/10.1007/s10973-018-7529-8>
- [11] Hwang, S.D., Kwon, H.G., Cho, H.H. (2010). Local heat transfer and thermal performance on periodically dimple-protrusion patterned walls for compact heat exchangers. *Energy*, 35(12): 5357-5364.
<https://doi.org/10.1016/j.energy.2010.07.022>
- [12] Zhao, Z., Xu, L., Gao, J.M., Xi, L., Ruan, Q.C., Li, Y.L. (2022). Multi-objective optimization of parameters of channels with staggered frustum of a cone based on response surface methodology. *Energies*, 15(3): 1240.
<https://doi.org/10.3390/en15031240>
- [13] Rao, Y., Xu, Y.M., Wan, C.Y. (2012). An experimental and numerical study of flow and heat transfer in channels with pin fin-dimple and pin fin arrays. *Experimental Thermal and Fluid Science*, 38: 237-247.
<https://doi.org/10.1016/j.expthermflusci.2011.12.012>
- [14] Kumar, G., Sharma, K.R., Dwivedi, A., Yadav, A.S., Patel, H. (2014). Experimental investigation of natural convection from heated triangular fin array within a rectangular enclosure. *International Review of Applied Engineering Research*, 4(3): 203-210.
- [15] Khaled, A.R.A., Gari, A.A. (2014). Heat transfer enhancement via combined wall and triangular rooted-fin system. *Journal of Electronics Cooling and Thermal Control*, 4: 12-21.
<https://doi.org/10.4236/jectc.2014.41002>
- [16] Dogan, M., Sivrioglu, M., Yilmaz, O. (2014). Numerical analysis of natural convection and radiation heat transfer from various shaped thin fin-arrays placed on a horizontal plate-a conjugate analysis. *Energy Conversion and Management*, 77: 78-88.
<https://doi.org/10.1016/j.enconman.2013.09.021>
- [17] Alsahy, M.J., Nashee, S.R., Ouda, A.A. (2025). Enhanced heat transfer performance in tubes using double-twisted tapes with integrated triangle winglets. *International Journal of Energy Production and Management*, 10(2): 333-342.
<https://doi.org/10.18280/ijepm.100214>
- [18] Aweda, F.O., Samson, T.K. (2024). Statistical analysis of wind speed characteristics using the Weibull distribution at selected African stations. *Journal of Sustainability for Energy*, 3(3): 187-197.
<https://doi.org/10.56578/jse030304>
- [19] Kumar, N., Puranik, B.P. (2017). Numerical study of convective heat transfer with nanofluids in turbulent flow using a Lagrangian-Eulerian approach. *Applied Thermal Engineering*, 111: 1674-1681.
<https://doi.org/10.1016/j.applthermaleng.2016.08.038>
- [20] Chen, Z.L., Luo, Y., Wang, Z.H., Liu, Y.L., Gai, L.M., Wang, Q.C., Hong, B.Y. (2024). Optimization design and performance study of a heat exchanger for an oil and gas recovery system in an oil depot. *Energies*, 17(11): 2631. <https://doi.org/10.3390/en17112631>
- [21] Rashid, F.L., Fakhruddin, S.K., Eleiwi, M.A., Hussein, A.K., Tahseen, T.A., Younis, O., Ahmed, M.I. (2022). CFD simulation in thermal-hydraulic analysis of airflow on different attack angles of row flat tube. *Frontiers in Heat and Mass Transfer*, 19: 1-8.
<https://doi.org/10.5098/hmt.19.6>
- [22] Wu, Z.J., You, S.J., Zhang, H., Zheng, W.D. (2020). Experimental investigation on heat transfer characteristics of staggered tube bundle heat exchanger immersed in oscillating flow. *International Journal of Heat and Mass Transfer*, 148: 119125.
<https://doi.org/10.1016/j.ijheatmasstransfer.2019.119125>
- [23] Mutasher, D.G. (2024). Advances in oscillating fin technologies for heat transfer enhancement: A review of numerical and experimental approaches. *Power Engineering and Engineering Thermophysics*, 3(4): 279-297. <https://doi.org/10.56578/peet030405>
- [24] Sivasankaran, S., Janagi, K. (2022). Numerical study on mixed convection flow and energy transfer in an inclined channel cavity: Effect of baffle size. *Mathematical and Computational Applications*, 27(1): 9.
<https://doi.org/10.3390/mca27010009>
- [25] Handoko, B.L., Indrawati, D.S., Zulkarnaen, S.R.P. (2024). Embracing AI in auditing: An examination of auditor readiness through the TRAM framework. *International Journal of Computational Methods and Experimental Measurements*, 12(1): 53-60.
<https://doi.org/10.18280/ijcmem.120106>
- [26] Al-Gaheeshi, A.M.R., Rashid, F.L., Eleiwi, M.A., Basem, A. (2023). Thermo-hydraulic analysis of mixed convection in a channel-square enclosure assembly with hemi-sphere source at the bottom. *International Journal of Heat and Technology*, 41(3): 551-562.
<https://doi.org/10.18280/ijht.410307>
- [27] Lindqvist, K., Skaugen, G., Meyer, O.H.H. (2021). Plate fin-and-tube heat exchanger computational fluid dynamics model. *Applied Thermal Engineering*, 189: 116669.
<https://doi.org/10.1016/j.applthermaleng.2021.116669>
- [28] Foual, M., Chaib, K., Chemloul, N.S., Becheffar, Y. (2024). Numerical study of the influence of amplitudes on heat transfer in a tube bundle. *International Journal of Modern Physics C*, 35(9): 2450111.
<https://doi.org/10.1142/S0129183124501110>
- [29] Afgan, I., Kahil, Y., Benhamadouche, S., Ali, M., Alkaabi, A., Berrouk, A.S., Sagaut, P. (2023). Cross flow over two heated cylinders in tandem arrangements at subcritical Reynolds number using large eddy simulations. *International Journal of Heat and Fluid Flow*, 100: 109115.
<https://doi.org/10.1016/j.ijheatfluidflow.2023.109115>
- [30] Hao, B., Gao, J.F., Guo, B.G., Ai, B.J., Hong, B.Y.,

- Jiang, X.S. (2022). Numerical simulation of premixed methane–Air explosion in a closed tube with U-Type obstacles. *Energies*, 15(13): 4909. <https://doi.org/10.3390/en15134909>
- [31] Eleiwi, M.A., Yassen, T.A., Khalaf, A.E. (2024). Improving the overall performance of a hybrid photovoltaic system by reflectors and cooling with water jet. *International Journal of Ambient Energy*, 45(1). <https://doi.org/10.1080/01430750.2024.2308757>
- [32] Ahmed, M.A., Alabdaly, I.K., Hatem, S.M., Hussein, M.M. (2024). Numerical investigation on heat transfer enhancement in serpentine mini-channel heat sink. *International Journal of Heat and Technology*, 42(1): 183-190. <https://doi.org/10.18280/ijht.420119>
- [33] Courand, A., Metz, M., Héran, D., Feilhes, C., Prezman, F., Serrano, E., Bendoula, R., Ryckewaert, M. (2022). Evaluation of a robust regression method (RoBoost-PLSR) to predict biochemical variables for agronomic applications: Case study of grape berry maturity monitoring. *Chemometrics and Intelligent Laboratory Systems*, 221: 104485. <https://doi.org/10.1016/j.chemolab.2021.104485>
- [34] Goldstein, R.J., Ibele, W.E., Patankar, S.V., Simon, T.W., et al. (2010). Heat transfer—A review of 2004 literature. *International Journal of Heat and Mass Transfer*, 53(21-22): 4343-4396. <http://doi.org/10.1016/j.ijheatmasstransfer.2010.05.004>
- [35] Menni, Y., Azzi, A., Zidani, C., Benyoucef, B. (2017). Effect of fin spacing on turbulent heat transfer in a channel with cascaded rectangular-triangular fins. *Journal of New Technology and Materials*, 7(2): 10-21.
- [36] Bhuvan, S., Kishore, P.S. (2024). Thermal performance of triangular fins on a computer processing unit. *International Advanced Research Journal in Science, Engineering and Technology*, 11(11): 48-57. <https://doi.org/10.17148/IARJSET.2024.111106>
- [37] Hithaish, D., Saravanan, V., Umesh, C.K., Seetharamu, K.N. (2022). Thermal management of electronics: Numerical investigation of triangular finned heat sink. *Thermal Science and Engineering Progress*, 30: 101246. <https://doi.org/10.1016/j.tsep.2022.101246>
- [38] Hameed, V.M., Hussein, M.A., Dhaiban, H.T. (2020). Investigating the effect of various fin geometries on the thermal performance of a heat sink under natural convection. *Heat Transfer*, 49(8): 5038-5049. <https://doi.org/10.1002/htj.21866>