IARJSET ISSN (O) 2393-8021, ISSN (P) 2394-1588

International Advanced Research Journal in Science, Engineering and Technology

Impact Factor 8.066 ⋇ Peer-reviewed / Refereed journal ⋇ Vol. 11, Issue 11, November 2024

DOI: 10.17148/IARJSET.2024.111106

THERMAL PERFORMANCE OF TRIANGULAR FINS ON A COMPUTER PROCESSING UNIT

Surisetty.Bhuvan [1*], P. S. Kishore [2]

M. Tech Student, Heat Power Engineering, Department of Mechanical Engineering,

Andhra University College of Engineering(A), Andhra University, Visakhapatnam 530003, India[1]

Department of Mechanical Engineering,

Andhra University College of Engineering(A), Andhra University, Visakhapatnam 530003, India[2]

Abstract: This study investigates the heat transfer performance of rectangular and triangular fins on a heat sink system, designed using real-time application parameters. The heat sink, consisting of 15 fins, which was modeled in CATIA software, focusing on the impact of varying fin lengths on performance parameter's such as heat flow rate, heat loss per unit mass, effectiveness, and efficiency. The research combines theoretical and calculations and Computational Fluid Dynamics (CFD) simulations in ANSYS Fluent to analyze fluid dynamics and heat transfer characteristics. The study compares the efficiency, rate of heat flow per unit mass and effectiveness of both fin shapes, Through the simulations, detailed insights into fluid flow and temperature distribution were gained, providing a comprehensive understanding of how geometry affects heat sink performance. The findings could aid in optimizing heat sink designs across industries like electronics, automotive, and aerospace, where efficient thermal management is essential for performance and reliability.

Keywords: Heat sink, thermal performance, fin shapes, CFD simulations and heat dissipation.

I INTRODUCTION

Electronics cooling is critical in ensuring the reliable operation which decides life-span of an electronic devices. As electronic components become more compact and powerful, they generate more heat during operation, if heat generated is not efficiently dissipated, which can lead to overheating and potentially cause failure. Effective thermal management in electronics helps to maintain optimal operating temperatures, preventing thermal stress that can damage sensitive components like semiconductors, integrated circuits, and processors. As the demand for faster, & more powerful electronics continues to grow, efficient cooling solutions become even more essential for maintaining device performance and preventing thermal-related failures.

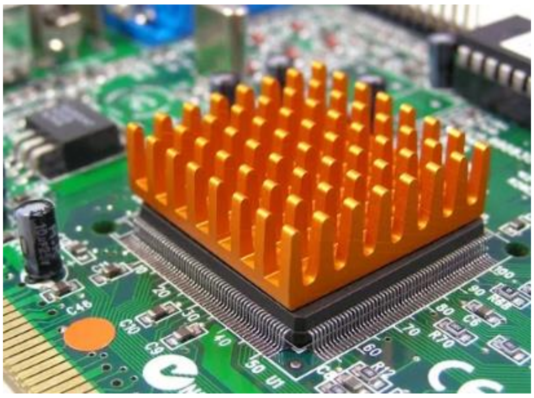


Fig.1: Heat sink for electronic cooling.

IARJSET ISSN (O) 2393-8021, ISSN (P) 2394-1588

International Advanced Research Journal in Science, Engineering and Technology

Impact Factor 8.066 ⫻ Peer-reviewed / Refereed journal ⫻ Vol. 11, Issue 11, November 2024

DOI: 10.17148/IARJSET.2024.111106

In modern electronics, maintaining optimal temperatures is essential for performance, reliability, especially in high-performance systems like computers, smartphones, and automotive electronics. Overheating can cause reduced speeds, malfunctions, and damage, leading to higher maintenance costs. Cooling methods like heat sinks, fans, and liquid cooling are crucial for dissipating heat and improving energy efficiency. As devices shrink, thermal management becomes more challenging. Advanced cooling technologies, such as optimized heat sink designs, nanofluids, and phase change materials, are being developed to meet these challenges, ensuring high-performance and energy-efficient electronics.

HEAT SINK:

A heat sink is a passive cooling device designed to dissipate heat from electronic components, such as CPUs, GPUs, or power transistors, ensuring they operate within optimal temperature ranges. Made from high thermal conductivity materials like aluminum or copper, heat sinks consist of a metal base and fins that increase the surface area for heat transfer. The heat sink absorbs heat from the component and transfers it to the fins, where it dissipates into the surrounding air through convection. A larger surface area enhances the heat sink's efficiency, which is why many designs incorporate thin, evenly spaced fins to maximize heat dissipation.

In high-performance electronic systems, heat sinks prevent thermal throttling, where components slow down to avoid overheating. They are often paired with fans to improve airflow and cooling efficiency. In extreme cases, liquid cooling systems may be used alongside heat sinks to manage higher heat loads. As electronic devices become smaller and more powerful, heat sink designs continue to evolve, incorporating innovations such as heat pipes, vapor chambers, and advanced materials to meet the increasing thermal demands of modern systems. A CPU heat sink with an integrated fan is a common solution in computers, where the fan enhances the heat sink's ability to dissipate heat from the CPU, ensuring stable performance in demanding environments.

PROBLEM IDENTIFICATION:

This study addresses the challenge of managing heat generated by high-performance CPUs, which can lead to overheating, reduced efficiency, and system failure, particularly in compact systems like laptops and gaming consoles. Traditional cooling methods, such as passive heat sinks, may not provide sufficient thermal dissipation for modern processors, especially as their power increases. The study uses computational fluid dynamics (CFD) simulations and analytical calculations to evaluate the thermal performance of CPU heat sinks with rectangular and triangular fins. By modeling these heat sinks in CATIA and performing simulations in ANSYS Fluent, the research investigates the impact of fin length, geometry, and configuration on heat dissipation and efficiency, aiming to optimize designs for better thermal management and improved system performance.

With the rapid advancement of electronic and mechanical devices, such as air conditioning systems, turbines, and electronic equipment, the need for efficient, compact, and lightweight heat sinks has become critical. Traditional cooling methods, like passive heat dissipation, are often insufficient for handling higher heat loads in modern, compact systems. To address this, researchers have focused on improving heat transfer rates using extended surfaces like fins, which increase the heat transfer area and enable more effective heat dissipation, making fin-based heat sinks a popular solution in high-performance thermal management.

LITERATURE REVIEW

Mao-Yu and Cheng-Hsiung [1] conducted an experimental and numerical study on the heat transfer rate of two pin fin heat sinks under natural convection. The study examined the effects of base plate characteristics, fin height, hole diameter, and porosity on heat transfer performance. Thermal performance improved with greater fin height, hole diameter, and input heat. The hollow heat sink outperformed the solid base design when its porosity was ≤ 0.262.

Sing [2] studied the thermal performance of a heat sink under natural convection using ANSYS software, focusing on circular pin fins with a length of 32 mm. The study varied the pin fin diameter by adjusting the angle of expansion to 1°, 2°, and 3°. suggesting that slight outward expansion enhances natural convection and improves heat transfer. This finding crucial for improving heat sink designs in various applications.

Effendi et al. [3] predicted the Nusselt number correction for a heat sink with round hollow hybrid fins (HHFHSs) under natural convection using CFD software. They studied 108 cases with different base temperatures (50°C, 70°C, and 90°C) and validated the numerical results through experiments. A Nusselt number correlation was developed based on parameters such as fin height, Rayleigh number, fin wall thickness, and external fin diameter. The correlation showed good accuracy, with less than a 20% difference compared to more complex numerical methods.

Mounika et al. (2016) [4] The heat transfer processes take place from the coolant to the tubes then from the tubes to the air through the fins. After the analysis is carried out, the heat transfer coefficient of air and ethylene glycol is estimated and further overall heat transfer coefficient is calculated.

DOI: 10.17148/IARJSET.2024.111106

Kishore.P.S et al. (2010) [5] The momentum and thermal eddy diffusivity characteristics are evaluated from the experimental heat transfer and pressure drop data. It is observed that the influence of nanoparticle in the base liquid water on the momentum eddy transport is not perceptible for the range of concentration considered. However, the presence of nanoparticles in water yielded enhancements in convective heat transfer. An improvised method for the evaluation of both eddy momentum and thermal diffusivities as a function of dimensionless velocity u+ and distance y+ is presented.
Baldry .M et al. (2019) [6] Aim of this study is to brief the previous investigation attempted enhancing the heat sinks thermal performance and to provide help to understand the cooling ability of their specific geometries. The various enhancement techniques used for optimizing the hydrothermal design of a pin fin, flat fin, micro-channel, and topology optimized heat sinks were summarized
PK Sarma et al. (2010) [7] The present investigation deals with a differential formulation to estimate the eddy diffusivity together with the universal velocity for fully developed turbulent flows in a tube. The subsequent theoretical predictions of wall friction coefficients and Nusselt numbers are in reasonable agreement with the classical solution of Blasius wall friction coefficient and Dittus and Boelter correlation for heat transfer, Nu=0.023 $Re^{0.8}$ $Pr^{1/3}$, The prediction of Nusselt numbers from the eddy diffusivity expression satisfactorily agree with the Dittus and Boelter heat transfer correlation.
Al-Damook.A et al (2015) [8] The aim of this study is to brief the previous investigation attempted enhancing the heat sinks thermal performance and to provide help to understand the cooling ability of their specific geometries. The various enhancement techniques used for optimizing the hydrothermal design of a pin fin, flat fin, micro-channel, and topology optimized heat sinks were summarized.
Kishore.P.S et al. 2001[9] The development of high-performance thermal systems has stimulated interest in methods to augment or intensify heat transfer rate. The performance of conventional heat exchangers can be substantially improved by a number of augmentation techniques. A good amount of research effort of both theoretical and experimental nature can be found in the literature defining the conditions under which an augmentation technique will improve heat and mass transfer rates.
Mao-Y W et al [10]. This paper presents a numerical simulation of the heat transfer performance under forced convection for two different types of circular pin fin heat sinks with (Type A) and without (Type B) a hollow in the heated base. COMSOL Multiphysics, which is used for the thermal hydraulic analyses, has proven to be a powerful finite-element-based simulation tool for solving multiple physics-based systems of partial and ordinary differential equations. The standard κ- ε two-equations turbulence model is employed to describe the turbulent structure and behavior.

II MODEL ANALYSIS OF TRIANGULAR FIN

1.Heat lost by fin, (Q_R)

$$\mathrm{KA}_c \mathrm{m}\theta_0 \frac{\mathrm{h\ cosh}ml + \mathrm{km\ sinh}\, ml}{\mathrm{Km\ cosh}\, ml + \mathrm{h\ sinh}\, ml}$$

Where,
k=thermal conductivity, W/mk

A_c = Cross section area of fin, m^2

m=fin parameter, $\left(\sqrt{hP/kA_c}\right)$

P=perimeter of fin, $(2W+4\delta), m$

θ_0 =Temperature difference, k

h = heat transfer coefficient, W/m^2K

Mass of rectangular fin, $(m_r) = 2\delta \times \rho \times L$

ρ =Density of fin material, kg/m^2

2.Rate of heat flow per unit mass through rectangular fin,

$$q_r = \frac{KA_C m\theta_0 \dfrac{h\cosh ml + km\sinh ml}{km\cosh ml + h\sinh ml}}{2\delta \times \rho \times L}$$

3.Efficiency of rectangular fin, (η_r)

$$= \frac{Heat\ lost\ with\ fin}{Heat\ lost\ from\ the\ fin}$$

IARJSET ISSN (O) 2393-8021, ISSN (P) 2394-1588

International Advanced Research Journal in Science, Engineering and Technology

Impact Factor 8.066 ⁙ Peer-reviewed / Refereed journal ⁙ Vol. 11, Issue 11, November 2024

DOI: 10.17148/IARJSET.2024.111106

If entire surface maintained at room temperature

$$=\frac{-KA_C m\theta_0 \frac{h\cosh ml+km\sinh ml}{km\cosh ml+h\sinh ml}}{2WLh\theta_0}$$

4.Effectiveness of rectangular fin, $\in_r = \frac{Heat\ lost\ with\ fin}{Heat\ loss\ without\ fin}$

$$=\frac{KA_C m\theta_0 \frac{h\cosh m\,l+km\sinh m\,l}{km\cosh m\,l+h\sinh m\,l}}{hA_C\theta_0}$$

1.Heat lost by triangular fin, $Q = 2W\theta_0\sqrt{(hk\delta)}\left[\frac{I_1(2B\sqrt{L})}{I_0(2B\sqrt{L})}\right]$

Where,

θ_0 = Temperature difference, K

k= thermal conductivity, W/Mk

B=fin parameter, $\left(\sqrt{hL/K\delta}\right)$

I_1=Bessel function of first kind

I_0=Bessel function of first kind

The mass of triangular fin, $(m_t) = \frac{1}{2}\times 2\delta\times L\times W\times\rho.$

ρ = Density of fluid kg/m^3

2.Rate of heat flow per unit mass, $(q) = \frac{\text{Heat flow through fin}}{\text{mass of fin}}$

$$=\frac{2W\sqrt{hk\delta}\theta_0\frac{I_1(2B\sqrt{L})}{I_0(2B\sqrt{L})}}{\frac{1}{2}2\delta\times L\times W\times\rho}$$

3.Efficiency of triangular fin $(\eta_r) = \frac{2W\sqrt{\delta\theta_0}\frac{I_1(2B\sqrt{L})}{I_0(2B\sqrt{L})}}{2WLh\theta_0}$

4.Effectiveness of triangular fin, $(\in_r) = \frac{Heat\ lost\ with\ fin}{Heat\ loss\ without\ fin}$

$$=\frac{2W\sqrt{hk\delta}\theta_0\frac{I_1(2B\sqrt{L})}{I_0(2B\sqrt{l})}}{hA_b\theta_0}$$

Where,

A_b =base area of triangular fin, $(W\times 2\delta)$, m^2

III COMPUTER AIDED MODELING

The heat sink presented in the image is designed in a rectangular shape, optimized for efficient thermal management. Its dimensions are carefully chosen, with a length of 31 mm, a width of 90 mm, and a depth of 42.9 mm, ensuring compatibility with the intended application. Each fin, a critical component of the heat sink, has a thickness of 0.9 mm, contributing to a balance between structural stability and maximum surface area for heat dissipation. The design process utilizes CATIA software, specifically employing the Part Design module, which provides advanced tools for precise modeling. By leveraging sketch, pad, and pocket commands, the design begins with a single fin, ensuring accuracy and uniformity. This fin is then patterned into multiple instances, creating a consistent array, while a 1 mm-thick base is added to serve as the foundation for the fins, ensuring structural integrity and optimal heat transfer to the surrounding environment.

IARJSET ISSN (O) 2393-8021, ISSN (P) 2394-1588

International Advanced Research Journal in Science, Engineering and Technology

Impact Factor 8.066 ⁙ Peer-reviewed / Refereed journal ⁙ Vol. 11, Issue 11, November 2024

DOI: 10.17148/IARJSET.2024.111106

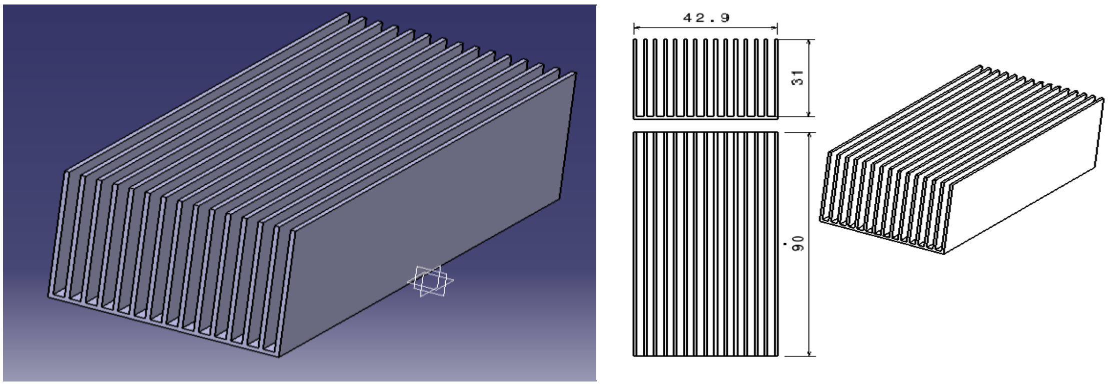


Fig.2: Rectangular heat sink.

The use of CATIA's Part Design module facilitates an efficient and streamlined workflow for the creation of the heat sink. Initially, the fin geometry is carefully sketched to match the required dimensions and thermal performance criteria. The pad command is used to extrude the fin geometry, creating the three-dimensional structure. To create a full array of fins, the pattern tool is employed, automating the duplication process and ensuring precise spacing between fins to maximize airflow and heat dissipation. The base of the heat sink is modeled separately, using similar commands to ensure compatibility with the fin array. This modular approach not only enhances design flexibility but also allows for easy modifications to accommodate specific application requirements.

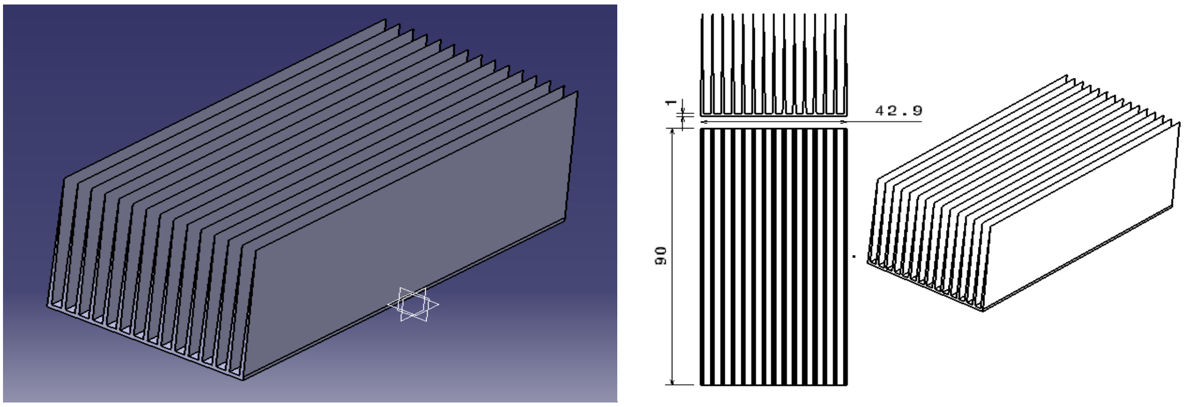


Fig.3: Triangular shaped fins in heat sink.

The heat sink shown in the image features a triangular design, tailored to maximize thermal dissipation while maintaining a compact form. With dimensions of 31 mm in length, 90 mm in width, and 42.9 mm in depth, it is structured to effectively dissipate heat for efficient cooling. Each fin, with a thickness of 0.9 mm, is crafted to balance durability and an expanded surface area for improved heat transfer. Designed using CATIA software's Part Design module, the process involved creating the geometry of a single fin through sketch and pad commands. The pocket command was used for precise modifications, ensuring the triangular shape adhered to the design specifications. This single fin was then replicated into a pattern to form the complete array, ensuring uniform spacing and consistency. The base of the heat sink, with a thickness of 1 mm, was modeled separately to provide structural support and ensure optimal contact with the heat source. The triangular shape and meticulously designed fins enhance airflow and heat dissipation, demonstrating an efficient thermal solution created through advanced CAD techniques. Upon completion of the design, the model was exported as a STEP file, allowing for its import into ANSYS Fluent for detailed simulation and analysis of fluid flow and thermal performance as shown in Fig.4 in detail.

Initially, the geometry is imported into the simulation software using the STEP file format. After importing the model, the geometry is checked for any errors or inconsistencies. These errors are identified and corrected using various tools available in the geometry cleanup process within the software. The final cleaned-up model consists of two distinct parts: the coil and the shell. This ensures that the model is accurate and ready for the next stages of simulation, where the detailed analysis of heat transfer and fluid flow will be conducted.

IARJSET ISSN (O) 2393-8021, ISSN (P) 2394-1588

International Advanced Research Journal in Science, Engineering and Technology

Impact Factor 8.066 ⁜ Peer-reviewed / Refereed journal ⁜ Vol. 11, Issue 11, November 2024

DOI: 10.17148/IARJSET.2024.111106

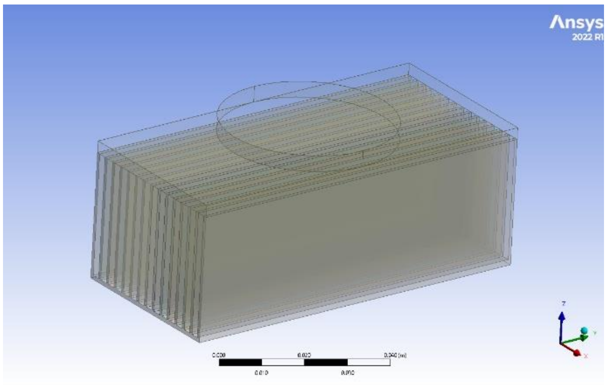


Fig.4: Geometry model of heat sink with air domain.

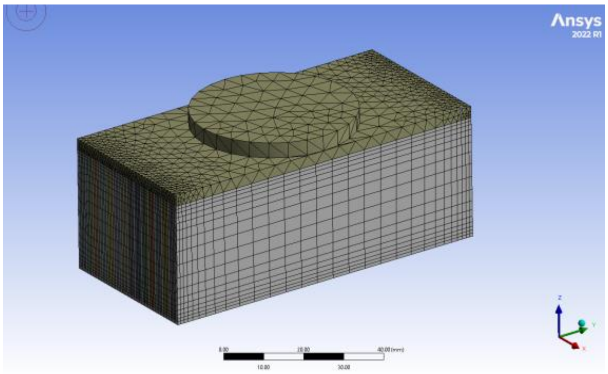


Fig.5: Meshed model of heat sink with air domain.

Meshing is crucial in Computational Fluid Dynamics (CFD) as it influences simulation accuracy and convergence. A high-quality mesh captures complex geometries and flow features more accurately, reducing errors and improving reliability. In Fig.5, tetrahedron elements were used for meshing due to their effectiveness in handling complex geometries. The assembly model was meshed with 87,693 cells and 79,235 nodes, providing a dense mesh that enhances the accuracy of fluid flow and heat transfer simulations.

In the CFD setup, boundary conditions define how fluids interact with the model. The inlet and outlet are labeled accordingly to ensure proper application of conditions at each fluid entry and exit point. The flow behavior is visualized through contour directions, which help analyze fluid dynamics and heat transfer across the mesh. In this setup, the inlet is positioned at the side of the domain to simulate real-world conditions, with air drawn in by the fan. The outlet is placed at the top of the domain, and the base temperature is applied to the heat sink's base.

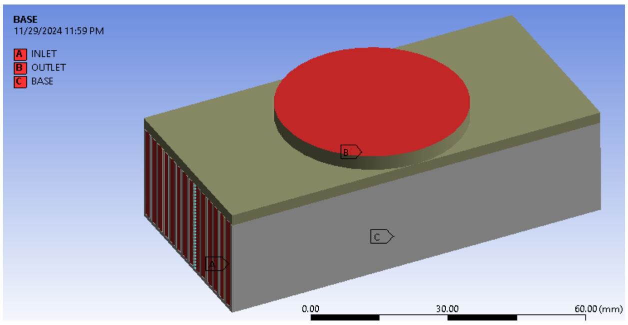


Fig.6: Boundary condition for heat sink model.

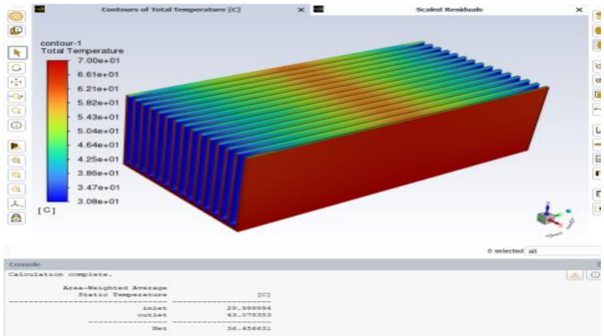


Fig.7: Temperature distribution for rectangular shape with 25 mm length.

In the next step, the solution phase begins by double-clicking on the solution node, which initiates the process of importing the geometry and finite volume data into ANSYS FLUENT software. Since the model is three-dimensional, FLUENT automatically recognizes and selects the 3D fluid dynamics option. This selection ensures that the software processes the data according to the three-dimensional geometry, allowing for accurate simulation of fluid flow and heat transfer within the complex model. This setup is crucial for performing detailed and reliable CFD analyses.

In ANSYS FLUENT, to complete the model solution, several essential steps must be followed. The turbulence model, k-epsilon, is selected for its robustness in simulating turbulent flow. Material properties, such as air for the fluid domain, are assigned, ensuring accurate simulation of the physical properties. Boundary conditions are defined to specify the interaction of the fluid with the model's boundaries, while reference and monitor conditions are set to track parameters during the solution. The solution method is chosen, followed by solution initialization to provide an initial guess for the solver. The model is then run to compute the fluid flow and heat transfer results for analysis.

IARJSET ISSN (O) 2393-8021, ISSN (P) 2394-1588

International Advanced Research Journal in Science, Engineering and Technology

Impact Factor 8.066 ⁒ Peer-reviewed / Refereed journal ⁒ Vol. 11, Issue 11, November 2024

DOI: 10.17148/IARJSET.2024.111106

For the analysis, the outlet condition is set to zero pressure, representing natural fluid flow from high to low pressure. This boundary condition helps simulate realistic flow behavior and pressure distribution across the heat exchanger. The interface region is considered a wall, preventing fluid flow inside it and allowing the calculation of heat transfer coefficients. Standard initialization from the inlet mass flow rate is used, allowing the setup to begin, including temperature, pressure, and velocity values. The analysis is run with convergence criteria of 10-5 for velocity, turbulence, and continuity, and convergence is achieved after 350 iterations for both models with and without coatings.

Fig.7 illustrates the temperature distribution for a rectangular heat sink with a length of 25 mm, showcasing how heat spreads across its surface during operation. The contour plot employs a color gradient, where red signifies the highest temperature, reaching 70°C, observed at the outer surfaces of the fins and base, which are in direct contact with the heat source. Conversely, regions where the fluid (air) interacts with the fins exhibit lower temperatures, highlighting the cooling effect of airflow. The air enters the system at an inlet temperature of 30°C and exits at 43.07°C, as depicted at the bottom of the contour plot. This temperature increase indicates the heat transfer from the heat sink to the air, facilitated by the fan's operation.

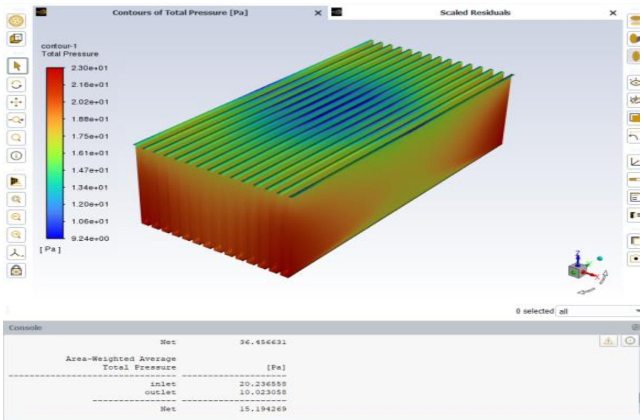


Fig.8: Pressure distribution for rectangular shape with 25 mm length.

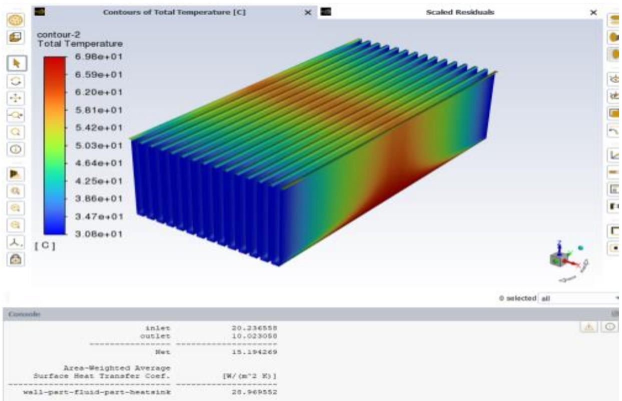


Fig.9: heat transfer coefficient for 25 mm rectangular fin.

Fig.8 illustrates the pressure distribution across a rectangular heat sink with a length of 25 mm, highlighting the areas of varying pressure intensities. The red color in the contour plot indicates regions of maximum pressure, primarily concentrated on the outer surfaces of the fins and the base where airflow encounters the greatest resistance. The average pressure readings are displayed below the contour plot, providing a clear indication of the pressure changes throughout the heat sink. At the inlet, the pressure is 20.23 Pa, while at the outlet, the average pressure drops to 10.02 Pa, reflecting the pressure loss as air flows through the fins. This pressure difference is a critical factor in understanding the aerodynamic performance of the heat sink, as it impacts the efficiency of airflow and, consequently, the overall heat dissipation capability of the System.

Fig.9 shows the temperature distribution across the heat sink, accompanied by the heat transfer coefficient values presented below the contour plot. The distribution highlights how temperature varies across different regions, with notable gradients indicating the efficiency of heat dissipation. The average heat transfer coefficient, which quantifies the rate of heat exchange between the heat sink and the surrounding air, is calculated to be 28.96 W/m²K. This value reflects the heat sink's effectiveness in transferring thermal energy from the fins and base to the air, facilitated by the design and airflow dynamics. The presented data is critical for evaluating the thermal performance of the heat sink and optimizing its design for improved cooling efficiency.

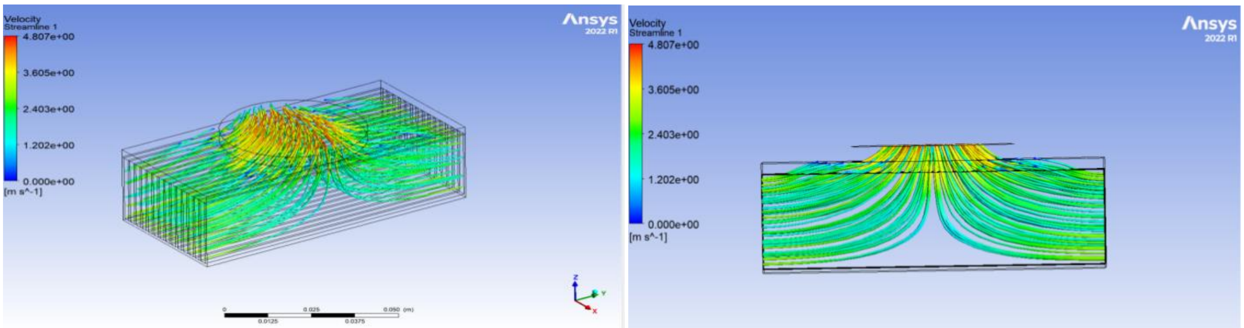


Fig.10: Velocity stream line distribution.

IARJSET ISSN (O) 2393-8021, ISSN (P) 2394-1588

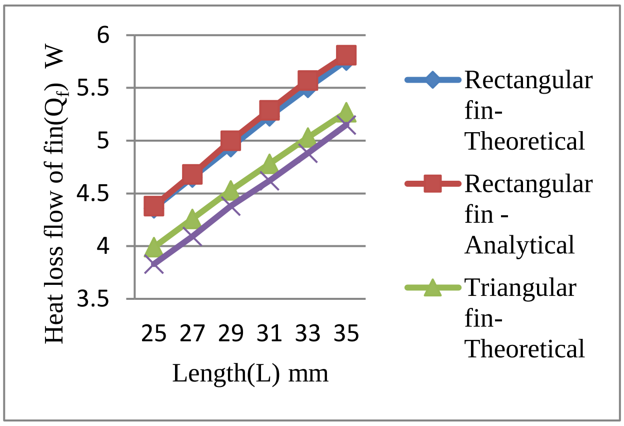
International Advanced Research Journal in Science, Engineering and Technology

Impact Factor 8.066 ⁒ Peer-reviewed / Refereed journal ⁒ Vol. 11, Issue 11, November 2024

DOI: 10.17148/IARJSET.2024.111106

Fig.10 Shows the velocity streamline distribution, showcasing the airflow dynamics as it moves from the inlet to the outlet of the system. The streamlines provide a visual representation of the flow path, indicating how air interacts with the heat sink structure. At the outlet, the air experiences compression and acceleration due to the flow constraints imposed by the fins and base geometry. This results in a maximum velocity of 4.8 m/s, highlighting areas of high kinetic energy. The streamline pattern and velocity variations are essential for understanding the efficiency of air circulation, which directly influences the cooling performance of the heat sink.

IV RESULTS AND DISCUSSION



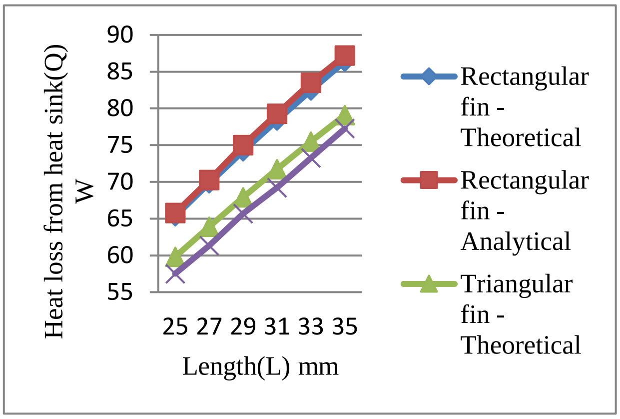
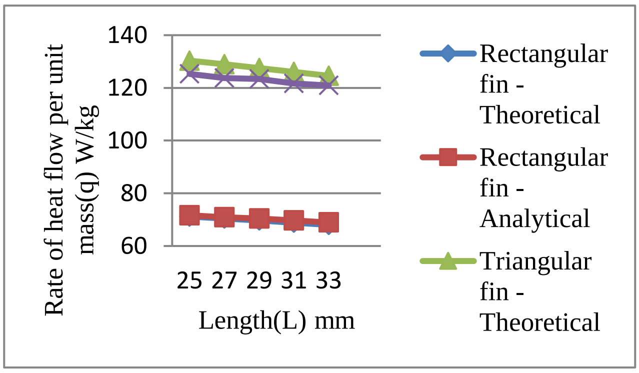


Fig.11: Variation of heat flow of the fin (Q_f) with length(L). Fig.12: Variation of Heat flow from Heat sink (Q)

Fig.11 shows the variation of heat flow through single fins of rectangular (Rec) and triangular (Tri) profiles at varying lengths using both analytical and CFD methods. Heat loss increases with fin length for both profiles, with rectangular fins showing higher heat loss, indicating better heat transfer. The analytical and CFD results closely match, with discrepancies under 1.5%. At 35 mm length, rectangular fins have 32.1% higher heat loss than at 25 mm, and triangular fins show similar increases. The triangular profile has slightly greater deviations between methods, especially at shorter lengths. While rectangular fins offer better performance, triangular fins are more compact and suitable for space-constrained applications.

Fig.12: Shows variation of Heat flow from a heat sink with 15 fins of rectangular (Rec) and triangular (Tri) profiles, based on analytical and CFD methods for fin lengths from 25 mm to 35 mm. Heat loss increases with fin length due to the larger surface area for heat dissipation, with rectangular fins showing consistently higher heat loss, indicating better thermal performance. The analytical and CFD results for rectangular fins differ by less than 1%, while the deviation for triangular fins ranges from 3.8% to 4.6%, highlighting the geometric complexity of triangular profiles. The heat loss for both profiles increases by around 32% with fin length, with rectangular fins offering higher efficiency. However, triangular fins provide a viable option for weight-sensitive applications, offering a balanced trade-off between performance and design constraints.



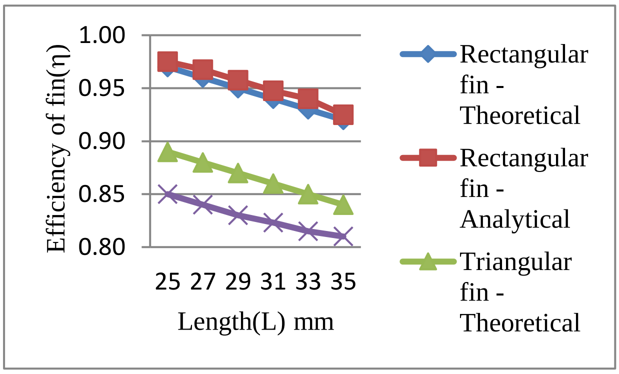


Fig.13: Variation of Rate of heat flow per unit mass of fin (q) with length(L).

Fig.14: Variation of efficiency (η) with length (L).

Fig.13: shows variation of Rate of heat flow rate per unit mass (W/kg) for rectangular (Rec) and triangular (Tri) fin profiles across varying fin lengths, using both analytical and CFD methods. For rectangular fins, heat flow decreases as

IARJSET ISSN (O) 2393-8021, ISSN (P) 2394-1588

International Advanced Research Journal in Science, Engineering and Technology

Impact Factor 8.066 ⁜ Peer-reviewed / Refereed journal ⁜ Vol. 11, Issue 11, November 2024

DOI: 10.17148/IARJSET.2024.111106

fin length increases, with analytical results dropping from 71.18 W/kg at 25 mm to 67.19 W/kg at 35 mm. CFD results show a similar decline of about 5.4%, as the increased mass of longer fins offsets the rise in heat dissipation. In contrast, triangular fins exhibit higher heat flow per unit mass, with analytical results decreasing by 5.6% from 130.33 W/kg at 25 mm to 123.03 W/kg at 35 mm, while CFD results show a 4.1% reduction. The triangular profile's superior performance is due to its lower mass, making it ideal for weight-sensitive applications, though CFD results for triangular fins show a slight deviation (up to 4.5%) from analytical predictions due to geometrical complexity. This highlights the trade-offs between thermal efficiency and weight in fin profile selection.

Fig.14: shows variation of efficiency for rectangular (Rec) and triangular (Tri) profiles at varying fin lengths, using both analytical and CFD methods. As fin length increases, efficiency decreases for both profiles due to greater thermal resistance along the length, reducing the temperature gradient and heat transfer effectiveness. For rectangular fins, efficiency drops from 0.97 at 25 mm to 0.91 (analytical) and 0.92 (CFD) at 35 mm, a 6.2% reduction. Triangular fins start with lower efficiency, at 0.89 (analytical) and 0.85 (CFD) for 25 mm, and reduce to 0.84 (analytical) and 0.82 (CFD) at 35 mm, reflecting a 5.6% (analytical) and 3.5% (CFD) decline.

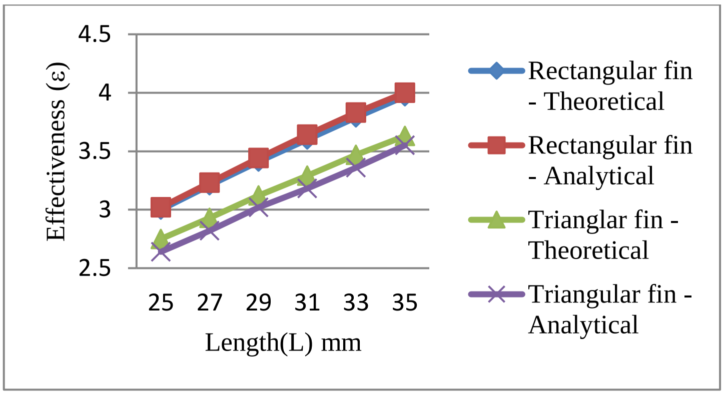


Fig.15: Variation of effectiveness (ε) with length (L).

Fig.15: shows variation of effectiveness of rectangular (Rec) and triangular (Tri) fins at varying lengths, assessed using both analytical and CFD methods. Effectiveness increases with fin length for both profiles due to a larger surface area for heat dissipation. For rectangular fins, effectiveness rises from 3.00 (analytical) and 3.02 (CFD) at 25 mm to 3.97 and 4.00 at 35 mm, a growth of about 32.3%. Triangular fins show a similar increase, with analytical results going from 2.75 to 3.63 and CFD from 2.64 to 3.55, a growth of 32% (analytical) and 34.5% (CFD). Rectangular fins have higher effectiveness due to their uniform cross-section, but the gap narrows for longer fins as triangular fins gain surface area.

V CONCLUSION

The study evaluated the performance of rectangular and triangular fins in a heat sink application, focusing on heat transfer, efficiency, and effectiveness at different lengths. Rectangular fins consistently outperformed triangular fins in heat dissipation, with up to 32% higher heat loss as fin length increased from 25 mm to 35 mm, due to their uniform cross-sectional area. While triangular fins offered better thermal efficiency per unit mass (5–6% higher), they had lower heat dissipation and slightly reduced efficiency. Both profiles showed similar trends in efficiency, with rectangular fins maintaining up to 97% efficiency and triangular fins 84–89%. The effectiveness of both fins increased with length, with rectangular fins showing a 32.3% improvement and triangular fins 34.5%. These findings highlight the trade-offs between thermal performance and weight/material optimization, with rectangular fins best for maximum heat dissipation and triangular fins suited for lightweight designs with moderate thermal needs.

NOMENCLATURE

A	Cross-sectional area, m^2
D	Diameter, m
h	heat transfer coefficient, W/m^2K
H	Height, m
$I_{1,0}$	Bessel function of First Kind
k	Thermal Conductivity, W/mK
L	Length, m
$m_{r,t}$	Mass of fin, kg
T	Temperature, k
W	Width, m
θ_0	Temperature difference, k

GREEK SYMBOLS

ρ	Density, kg/m^3
ε	Effectiveness
η	Efficiency

DOI: 10.17148/IARJSET.2024.111106

P	Perimeter, m
Q	Heat Transfer Rate, W
q	Rate of heat flow per unit mass, W/kg

REFERENCES

[1]. Mao-Y W., Cheng-Hsiung Y, "Natural convective performance of perforated heat sinks with circular pin fins", Journal of Heat and Mass Transfer, Vol.51, pp.1383-1392,2015.

[2]. Singh G, "Natural convection heat transfer from modified 1-degree, 2-degree, and 3-degree outward expansion of pin fins", International Journal of Engineering Science and Computing, Vol.7, pp.14567-14570,2017.

[3]. Effendi N.S., Severianus S.G., Putra R., Kim K.J, "Prediction methods for natural convection around hollow hybrid fin heat sinks", International Journal of Thermal Sciences, Vol.126, pp.272–280,2018.

[4] P. Mounika, Rajesh K Sharma, P. S. Kishore, "Performance Analysis of Automobile Radiator", International Journal of Recent Technologies in Mechanical and Electrical Engineering, Vol.3, PP 35-38, 2016.

[5]. PK Sarma, Kedarnath Chada, KV Sharma, L Shyam Sundar, PS Kishore, V Srinivas, "Experimental study to predict momentum and thermal diffusivities from convective heat transfer data of nano fluid with Al2O3 dispersion", International Journal of Heat and Technology, Vol.28, PP 123-131, 2010.

[6]. Baldry M., Timchenko V., Menictas C, "Optimal design of a natural convection heat sink for small thermoelectric cooling modules", Applied Thermal Engineering, Vol.160, pp.1-13,2019.

[7] PK Sarma, Kedarnath Chada, V Dharma Rao, P.S. Kishore, T Subrahmanyam, AE Bergles, "Evaluation of momentum and thermal eddy diffusivities for turbulent flow in tubes", International Journal of Heat And Mass Transfer, Vol.53, PP 1237-1242, 2010.

[8]. Al-Damook A., Kapur N., Summers J.L., "Thompson H.M. An experimental and computational investigation of thermal air flows through perforated pin heat sinks", Journal of Applied Thermal Engineering, Vol.89, pp.365–376,2015.

[9] P S Kishore., "Experimental and theoretical studies of convective momentum and heat transfer in tubes with twisted tape inserts", Ph.D. Thesis Andhra University, Visakhapatnam, 2001.

[10]. Mao-Y W., Cheng-Hsiung Y., "Numerical study of thermal performance of perforated circular pin fin heat sinks in forced convection", Journal of Heat and Mass Transfer, Vol.53, pp.2031-2044,2017.