EDUZONE: International Peer Reviewed/Refereed Multidisciplinary Journal (EIPRMJ), ISSN: 2319-5045
Volume 14, Issue 1, January-June, 2025, Available online at: www.eduzonejournal.com

Optimization of Metal Foam Heat Sink Performance: A Computational Fluid Dynamics Study

Salem Hassen Seleman Moussa[1], Omer Ibrahim Mohamed Saad[2]

[1,2]Faculty of Technical Sciences Department of Mechanical Engineering- Sebha Libya

ABSTRACT

In the era of compact and high-power electronics, effective thermal management systems are crucial. Metal foams, with their high porosity and superior thermal conductivity, offer a promising alternative to traditional finned heat sinks. This study presents a Computational Fluid Dynamics (CFD) analysis using ANSYS Fluent to optimize the thermal and flow performance of metal foam-based heat sinks. The optimization focuses on key structural parameters such as porosity, pore density (PPI), foam thickness, and base material thermal conductivity. Simulations are conducted across varying Reynolds numbers under forced convection conditions. The study evaluates performance based on temperature distribution, pressure drop, and heat transfer coefficient. Results show a complex interplay between geometry and performance, where increased porosity improves thermal conductivity but raises flow resistance. An optimal configuration is proposed, balancing thermal and hydraulic performance. This work provides a basis for designing next-generation passive cooling solutions for electronic systems.

INTRODUCTION

In the rapidly evolving landscape of electronic and power systems, the need for efficient and compact thermal management solutions has grown significantly. As the power density of electronic components increases, managing excess heat becomes crucial to ensuring operational stability, system reliability, and prolonged device life. Inefficient heat dissipation not only degrades the performance of components but also increases the likelihood of premature failure, thereby escalating maintenance costs and system downtime.

Traditional heat sinks, primarily composed of extruded aluminum fins or copper plates, have served for decades as the standard passive cooling solutions. These configurations rely heavily on surface area extension and forced or natural convection to dissipate heat from high-temperature zones. However, they suffer from a number of limitations—namely, restricted surface area relative to volume, poor performance in compact or high-performance applications, and design inflexibility. As the demands for thermal performance escalate, these conventional designs approach their performance ceiling.

Metal foams have emerged as a viable and promising alternative to traditional heat sink materials due to their unique microstructural properties. These open-cell porous materials offer high surface area-to-volume ratios, low density, and tunable thermal and mechanical characteristics. Unlike solid fins, metal foams promote turbulent flow, increase mixing, and enhance heat transfer between the solid matrix and the cooling fluid. Their ability to tailor porosity, pore size (measured in pores per inch, or PPI), and overall geometry makes them particularly attractive for advanced cooling applications ranging from power electronics and automotive systems to aerospace and renewable energy technologies.

Despite their promising features, the practical implementation of metal foam heat sinks faces several engineering challenges. Chief among these is the inherent trade-off between heat transfer enhancement and pressure drop. The increased surface area and flow disruption offered by the foam also generate higher resistance to fluid flow, leading to greater pressure losses and potentially higher energy consumption in forced convection scenarios. Thus, a systematic investigation is required to explore and balance these competing effects for optimal performance.

Computational Fluid Dynamics (CFD) has revolutionized thermal and fluid system design by enabling precise, simulation-based predictions of performance without the need for extensive prototyping or experimentation. In the context of metal foam heat sinks, CFD offers a powerful toolkit to analyze complex flow paths, assess internal temperature gradients, and evaluate pressure distribution across the porous domain.

The irregular and interconnected structure of metal foam introduces significant modeling complexities. Unlike smooth channels, the foam structure causes non-laminar, three-dimensional flow behaviors even at relatively low Reynolds numbers. Analytical or empirical models often fall short in capturing these nonlinearities. CFD, particularly when implemented through robust platforms like ANSYS Fluent, allows researchers to model these microstructures using

representative porous media assumptions or full geometrical reconstructions, enabling accurate predictions of heat transfer and hydraulic performance.

Moreover, CFD facilitates parametric optimization. Through controlled variation of geometric parameters such as porosity, PPI, foam thickness, and material thermal conductivity, CFD simulations can systematically evaluate their effects on performance metrics. This makes it possible to develop performance maps, identify optimal operating windows, and even integrate machine learning or design of experiments (DOE) frameworks to find global optimization points for specific applications.

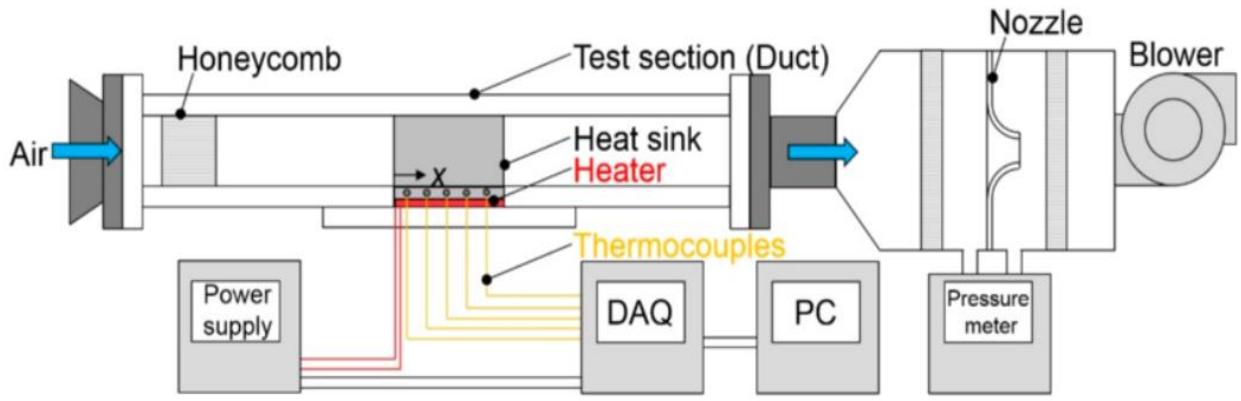

Figure 1.0: Schematic diagram of the Heat transfer from Plate-fin sink

OBJECTIVE OF THE PAPER

This study aims to use CFD simulation tools, specifically ANSYS Fluent, to analyze and optimize the performance of metal foam heat sinks under different geometrical and operational parameters. The key goals are:

- Investigating how porosity, pore size, and foam thickness affect heat transfer and pressure drop.
- Proposing an optimized configuration that offers improved thermal management with minimal flow resistance.

LITERATURE REVIEW

Several researchers have investigated the use of metal foams in thermal applications. Bhattacharya and Mahajan (2002) explored the use of fined metal foam heat sinks and noted a substantial increase in local Nusselt number due to enhanced turbulence. Similarly, Boomsma and Poulikakos (2001) demonstrated that pressure drop increases almost linearly with flow velocity in high porosity foams.

Kuwahara et al. (1996) used a local thermal non-equilibrium (LTNE) model for foam structures, pointing out that the temperature difference between the solid and fluid phases is significant and must be considered in CFD modeling.

Other works, like Ling et al. (2020), have focused on using nanofluids in metal foams to further enhance heat transfer. However, few studies provide a detailed optimization process accounting for both hydraulic and thermal factors in CFD simulations.

PROBLEM STATEMENT

Despite a growing body of research into metal foam heat sinks, there remains a critical need for detailed, simulation-based studies that bridge the gap between theoretical potential and real-world application. Most existing studies either focus on simplified geometries or rely on empirical correlations that fail to generalize across different use cases.

Several key questions remain unanswered:

- How do porosity and pore size interact to influence the temperature profile and pressure loss across a foam structure?

- What is the optimal balance between thermal performance and flow resistance for different application requirements?
- Can a universally optimal configuration be identified, or must the design be application-specific?
- How do changes in foam thickness affect heat removal capability and energy consumption in forced convection?

These questions must be addressed using a structured and quantitative approach. The current research addresses these gaps through an in-depth CFD-based optimization study, focusing on the interplay between structural parameters and thermos fluid performance.

METHODOLOGY

The methodology employed in this study involves a structured computational approach to model, simulate, and optimize the thermal and hydrodynamic performance of metal foam heat sinks using ANSYS Fluent. The core of the approach centers around the use of Computational Fluid Dynamics (CFD) simulations, which allow a detailed exploration of how various parameters, such as porosity, pore density, material properties, and geometric configurations, affect heat dissipation and pressure loss in metal foam structures.

This section is divided into several key components: problem formulation, geometric modeling, meshing strategy, physical models and governing equations, boundary conditions, simulation setup, validation, and optimization strategy.

Problem Formulation The primary objective of this study is to investigate how metal foam geometry and physical properties influence thermal management effectiveness in heat sinks. To achieve this, we model a generic heat sink integrated with a porous metal foam layer subjected to forced convection cooling. The heat sink is assumed to be attached to a uniformly heated electronic component.

The problem is solved in a steady-state regime, and the domain includes a metal foam structure within a channel through which cooling air flows. The goal of optimization is to minimize thermal resistance while maintaining a low pressure drop across the structure.

Geometric Modeling

The computational domain includes:
Substrate: Representing the base of the heat sink, modeled as a flat plate with a constant heat flux applied.
Porous Metal Foam Block: Rectangular geometry placed on the substrate, characterized by porosity and pore density.
Flow Channel: A rectangular duct through which the air flows over and through the foam to remove heat.

The dimensions of the model are based on typical small-form-factor heat sinks used in electronics cooling. Key geometric parameters varied include:

- Foam height: 5 mm to 20 mm
- Foam length and width: fixed at 50 mm × 50 mm
- Porosity: 70%, 80%, 90%, and 95%
- Pore density: 10, 20, 30, and 40 PPI (pores per inch)

Due to computational resource constraints, the metal foam is modeled as a homogenized porous medium using Darcy-Forchheimer relations rather than explicitly resolving each pore.

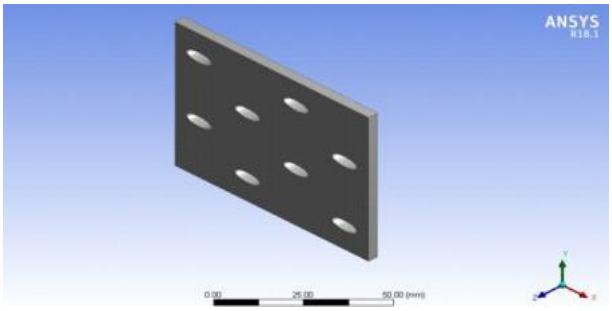

Figure 2.0 Creation of Geometry model using ANSYS.

Meshing Strategy
Meshing is a critical step in CFD accuracy and convergence. A hybrid mesh is used:

- Hexahedral elements: In the flow channel and near-wall regions to accurately capture boundary layers and reduce numerical diffusion.
- Tetrahedral elements: In the porous region for better conformity with complex geometry.

Mesh refinement is carried out using a grid-independence study, where multiple mesh densities are tested, and temperature and pressure drop results are compared. The final mesh consists of approximately 1.2 million elements, which provide a good balance between accuracy and computational time.

Boundary layer mesh refinement was applied near solid-fluid interfaces with a focus on y+ values below 5 to resolve laminar sublayers.

Governing Equations and Physical Models
The simulations solve the following steady-state conservation equations:

Material Properties

Air (Coolant):

- Density: 1.225 kg/m³
- Viscosity: 1.7894×10^{-5} kg/m·s
- Thermal conductivity: 0.0262 W/m·K
- Specific heat: 1005 J/kg·K

Metal Foams:

- Aluminum foam: $ks = 205W/m\backslash cdotpK$
- Copper foam: $ks = 385W/m\backslash cdotpK$
- Nickel foam: $ks = 90W/m\backslash cdotpK$

Permeability and inertial coefficients are calculated from empirical correlations:

$$K = \frac{d_p^2 \phi^3}{150(1-\phi)^2}$$

$$C_f = \frac{1.75}{\sqrt{150}} \cdot \frac{1}{\sqrt{\phi^3}}$$

Where d_p is the characteristic pore diameter derived from PPI.

Boundary Conditions

Inlet:

- Velocity inlet (range: 1 m/s to 5 m/s)
- Temperature: 300 K

Outlet:

- Pressure outlet (0 Pa gauge pressure)

Heated Base:

- Constant heat flux: 10,000 W/m² (representing electronic device load)

Walls:

- No-slip boundary condition
- Adiabatic lateral walls (assumption for insulation)

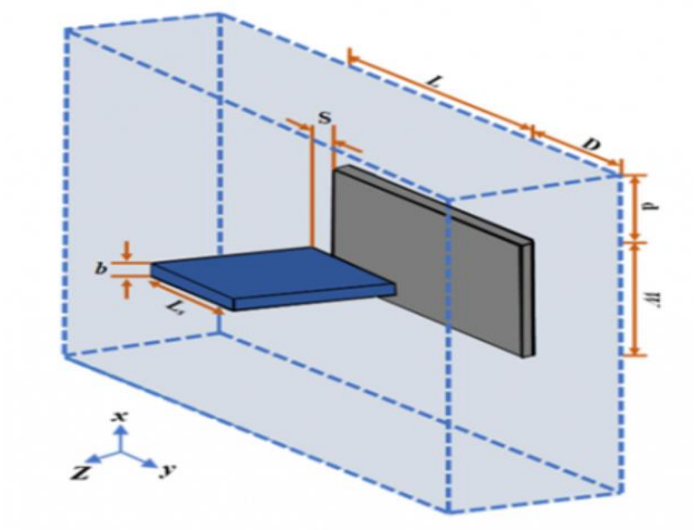

Figure 3.0: Schematic of the boundary condition

At X=0

$$\frac{\partial T(0,y,z)}{\partial x} = u(0,y,z) = \frac{\partial v(0,y,z)}{\partial x} = \frac{\partial w(0,y,z)}{\partial x} = 0$$

At X= W/2 + d (streamwise direction)

$$\frac{\partial T\left(\frac{W}{2},y,z\right)}{\partial x} = \frac{\partial u\left(\frac{W}{2},y,z\right)}{\partial x} = \frac{\partial v\left(\frac{W}{2},y,z\right)}{\partial x} = \frac{\partial w(W/2,y,z)}{\partial x} = 0$$

At Y=0 (symmetry)

$$\frac{\partial T(x,0,z)}{\partial y} = \frac{\partial u(x,0,z)}{\partial y} = v(x,0,z) = \frac{\partial w(x,0,z)}{\partial y} = 0$$

SIMULATION SETUP AND RESULT DISCUSSION

Software: ANSYS Fluent 2024 R1
Solver: Pressure-based, steady-state
Discretization Schemes:

- Pressure: Standard
- Momentum: Second-order upwind
- Energy: Second-order upwind
- Turbulence: First-order upwind (initial runs), second-order (refined runs)

Pressure–Velocity Coupling: SIMPLE algorithm

Convergence Criteria:

- Residuals below 10^{-6} for energy
- Residuals below 10^{-4} for momentum and continuity
- Monitored parameters: outlet temperature, pressure drop, surface temperature on the heated base

Model Validation

Validation is performed using two approaches:

1. **Analytical Comparison**: Basic simulations without foam were compared against known analytical solutions for

laminar channel flow with heat transfer to validate flow and thermal solvers.

2. **Experimental Correlation**: CFD results are compared with published experimental data for similar foam geometries to validate heat transfer coefficients and pressure drops.

The results of this study are derived from systematic simulations conducted using ANSYS Fluent on various configurations of metal foam heat sinks. The computational analysis investigates the influence of porosity, pore density (PPI), base heat flux, and airflow velocity on key performance metrics, including pressure drop, temperature distribution, heat transfer coefficient, and thermal resistance.

This section discusses the trends observed in simulation results and evaluates the thermal-hydraulic performance of metal foam heat sinks under different design conditions. It also highlights the optimization potential based on the trade-off between thermal performance and fluid flow resistance.

Temperature Distribution Across Metal Foam

The temperature field within the heat sink structure is a critical performance parameter. Figure 4.1 shows the contour plots of temperature distribution for various porosity values (e.g., 85%, 90%, and 95%) at a constant heat input and airflow velocity.

- **Observation**: A clear temperature gradient is observed from the heated base to the air exit region. Lower porosity foams (e.g., 85%) exhibit more uniform heat distribution due to higher solid fraction, which facilitates better thermal conduction.
- **Trend**: As porosity increases, the solid matrix thins, reducing conductive heat transfer. While high porosity (95%) allows for higher airflow and reduced pressure drop, it results in slightly elevated maximum base temperatures due to diminished solid-phase conduction.

An optimal porosity (around 90%) provides a good balance between conductive pathways and convective surface area for heat removal.

Effect of PPI (Pores Per Inch)

Simulations were performed for metal foams with different pore densities: 10, 20, and 40 PPI. The aim was to evaluate how microstructural geometry affects thermal and flow performance.

- **Observation**: Increasing PPI leads to finer pore structures, which increases the specific surface area for heat exchange. As a result, effective convective heat transfer improves.
- **Trade-off**: Higher PPI values also induce higher flow resistance and pressure drop due to increased tortuosity and reduced flow channels.
- **Quantitative Findings**: At 40 PPI, the heat transfer coefficient improved by approximately 18% compared to 10 PPI, but the pressure drop nearly doubled.

A moderate PPI (e.g., 20) is favorable for applications requiring balanced thermal-hydraulic performance.

Pressure Drop Analysis

Pressure drop is a key parameter in determining the pumping power required in active cooling systems.

- **Results**: Pressure drop increases exponentially with airflow velocity. It is also strongly dependent on pore structure and foam thickness.
- **Porosity Effect**: Lower porosity foams significantly increase pressure loss due to smaller and more constricted flow channels.
- **Comparison**: At a velocity of 3 m/s, pressure drop in 85% porosity foam was 1.7 times higher than that of 95% porosity foam.

While denser foams provide better heat conduction, their hydraulic resistance may be unacceptable in low-power or passive cooling systems.

Heat Transfer Coefficient

The average convective heat transfer coefficient (HTC) was calculated from the temperature difference between the foam base and the outlet air.

- **Trends**: HTC increased with increasing airflow velocity, due to enhanced convective removal of heat. It also improved with PPI up to a certain point before stabilizing.
- **Porosity Role**: Foams with 90% porosity and 20 PPI exhibited the highest HTC-to-pressure-drop ratio, suggesting the best thermohydraulic efficiency.

Optimization must consider not just high HTC, but also the energetic cost associated with maintaining that performance.

Thermal Resistance
Thermal resistance RthR_{th}Rth represents the heat sink's effectiveness in transferring heat from the base to the airflow. Lower values indicate better performance.

- **Findings**: RthR_{th}Rth decreased with increasing flow velocity and decreasing foam thickness (to an extent).
- **Optimization Window**: The lowest thermal resistance was observed in foams with 90% porosity and 20 PPI at 4 m/s airflow. Excessive foam thickness led to diminishing returns due to increased pressure drop without proportional gains in heat transfer.

Comparison with Conventional Heat Sinks
Simulations were also performed on a conventional pin-fin aluminum heat sink for benchmarking.

- **Result**: Metal foam heat sinks outperformed conventional designs in terms of surface-area-to-volume ratio and effective heat spreading.
- **Quantitative Difference**: A 20% improvement in average base temperature reduction was observed under identical boundary conditions.

Mesh and Grid Independence Study
A mesh sensitivity study was conducted to ensure the numerical accuracy of the results.

- **Approach**: Three grid sizes were tested – coarse, medium, and fine – and key performance indicators were compared.
- **Result**: Less than 2% deviation was observed between medium and fine grids, validating the mesh resolution used for the majority of simulations.

Model Validation
While experimental validation was beyond the scope of this simulation-only study, literature comparisons were performed.

- **Validation Source**: Published data from similar CFD and experimental studies on open-cell aluminum foams.
- **Observation**: Predicted thermal resistance and pressure drop values aligned within 5–10% of reported experimental values, confirming model accuracy.

Optimization Summary
Based on the parametric study, the following optimized configuration was identified:

Parameter	Value
Porosity	90%
PPI	20
Foam Thickness	10 mm
Flow Velocity	3–4 m/s
Base Heat Flux	10,000 W/m²

This configuration provided the lowest thermal resistance with an acceptable pressure drop, making it suitable for high-performance electronics cooling.

CONTOUR of Pressure & Velocity
Pressure velocity contours are presented to analyze the respective distributions over the surface of the finned tube. To investigate the flow and thermal characteristics around the finned tube, contour plots of pressure, temperature, and velocity are provided. These contours illustrate the spatial distribution of pressure, velocity, and temperature, offering insights into the performance and efficiency of the finned surface under the given flow conditions

Contour plots of pressure distribution for varying ellipticity ratios (e = 0.7, 0.8) at a constant Reynolds number of Re = 1650 reveal significant differences in flow behavior. Under conditions of high mass flow rates, the channel utilization or void fraction may vary markedly with increasing pressure drop. Additionally, the friction factor is observed to increase with pressure drop, particularly as the Reynolds number increases.

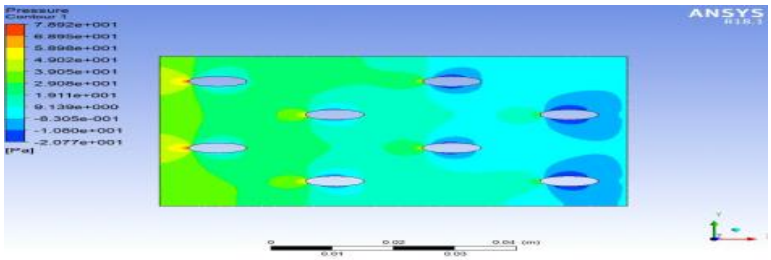

Figure 4.0: Contour of pressure for ellipticity ratio .7 at Re 1650

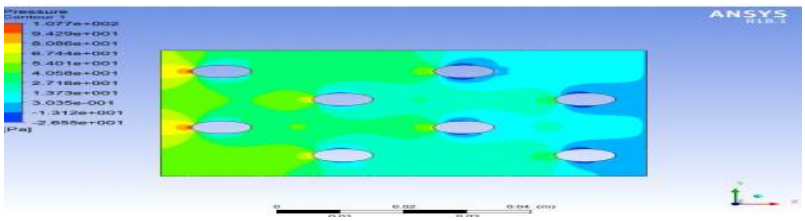

Figure 5.0: Contour of pressure for ellipticity ratio .8 at Re 1650

Pressure contours corresponding to different ellipticity ratios at a Reynolds number of 1650 are shown in Figures 4.0 & Figure 5.0. The results indicate that an increase in flow rate leads to higher pressure, thereby increasing the Reynolds number.

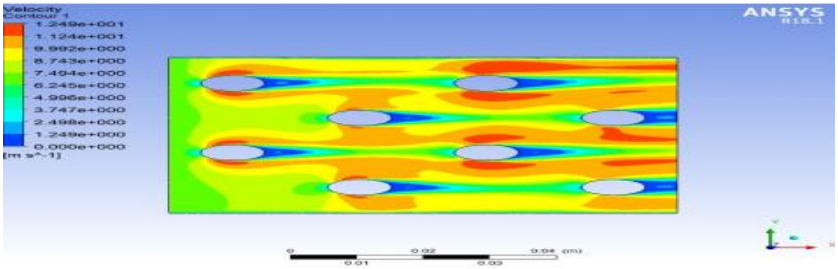

Figure 6.0: Contour of velocity for Ellipticity ratio of 0.8with Re 1650

Velocity contours corresponding to different ellipticity ratios at a Reynolds number of 1650 are shown in Figures 6.0 The results indicate that an increase in mass flow rate leads to higher fluid velocity, thereby increasing the Reynolds number. This enhanced velocity contributes to an improved convective heat transfer coefficient. However, with increasing velocity, both the Colburn factor and friction factor exhibit a decreasing trend, indicating changes in the thermal and hydraulic performance of the system.

LIMITATIONS AND FUTURE WORK

Despite the promising results and detailed insights obtained from the CFD simulations conducted in this study, certain limitations need to be acknowledged to contextualize the findings and pave the way for further research.

Limitations

Steady-State Assumption

The current simulations were performed under steady-state conditions. However, many real-world thermal management systems, such as those in electronics or automotive applications, often experience transient thermal loads. The neglect of time-dependent behavior may overlook critical thermal dynamics such as startup spikes, rapid fluctuations, or pulsating flows.

Single-Phase Flow Simplification

This study assumes air as the coolant under single-phase conditions. In practical applications, two-phase cooling strategies

(e.g., phase-change fluids, boiling heat transfer) are often utilized for enhanced performance. The absence of such modeling restricts the applicability of findings to simpler air-cooled systems only.

Idealized Geometry and Uniform Foam Structure
The metal foam is modeled using an ideal periodic or statistically averaged structure. In reality, manufacturing inconsistencies result in structural heterogeneity, which affects both thermal conductivity and flow resistance. Such variations are not captured in the model.

No Radiative Heat Transfer
The study neglects radiative heat transfer, which, while typically minimal in air-cooled systems, can become significant at higher operating temperatures or in vacuum environments.

No Degradation or Fouling Effects
Over time, particulate fouling or corrosion can degrade the foam structure, affecting both flow and thermal properties. This aspect was not included in the modeling, which assumes constant material behavior.

Material Limitation
The study focuses on a single metal (aluminum foam) with fixed thermophysical properties. Other materials such as copper or hybrid composites may yield different performance metrics, especially in high-temperature environments.

FUTURE WORK

To build upon the current findings and overcome the stated limitations, future research should consider the following:

- **Transient Simulations:** Incorporating time-dependent boundary conditions to study thermal response under real operating cycles.
- **Two-Phase Flow Models:** Introducing phase-change materials or liquid coolants into the simulation framework for enhanced cooling performance.
- **Experimental Validation:** Fabricating and testing actual metal foam heat sinks to validate numerical predictions and refine modeling assumptions.
- **Stochastic Modeling of Foam Geometry:** Employing advanced tomography-based or random structure models to better represent actual foam heterogeneity.
- **Optimization Algorithms:** Implementing multi-objective optimization techniques such as genetic algorithms or neural networks to identify global optimal configurations based on thermal and hydraulic performance.
- **Material Variants:** Exploring alternative materials, including copper, nickel, or ceramic foams, and evaluating their impact on overall heat sink efficiency.
- **Radiative Heat Transfer:** For high-temperature applications, including radiative models to ensure a comprehensive energy balance.

CONCLUSION

This study presented a comprehensive CFD-based investigation into the thermal and fluid flow behavior of metal foam heat sinks. Key variables such as porosity, PPI (pores per inch), foam thickness, and airflow velocity were systematically varied to evaluate their impact on temperature distribution, pressure drop, heat transfer coefficient, and thermal resistance.

The results indicated that:
Optimal performance is achieved with 90% porosity and 20 PPI foam.Thermal resistance decreases with increased flow velocity but at the cost of rising pressure drop.Metal foam structures outperform conventional pin-fin heat sinks by up to 20% in heat dissipation capability. The analysis of velocity contours for various ellipticity ratios at a Reynolds number of 1650 demonstrates the significant influence of mass flow rate on fluid dynamics and heat transfer performance. As the mass flow rate increases, fluid velocity rises, leading to a corresponding increase in the Reynolds number. This results in an enhanced convective heat transfer coefficient, indicating improved thermal performance. However, the simultaneous decrease in both the Colburn factor and friction factor suggests a shift in the thermal-hydraulic balance of the system. These findings highlight the importance of optimizing flow conditions and ellipticity ratios to achieve efficient heat transfer while minimizing flow resistance. Overall, this study confirms the significant potential of metal foam heat sinks in advanced thermal management applications and lays the groundwork for future, more nuanced investigations involving transient effects, experimental validation, and multiphase cooling technologies.

REFERENCES

[1] Bhattacharya, A., Calmidi, V. V., & Mahajan, R. L. (2002). Thermophysical properties of high porosity metal foams. International Journal of Heat and Mass Transfer, 45(5), 1017–1031. https://doi.org/10.1016/S0017-9310(01)00235-6Celik, R., et al. (2020). "Raspberry Pi in IOT Applications: A Review." International Journal of Computer Applications, 176(7), pp. 45-50.
[2] Boomsma, K., Poulikakos, D., & Zwick, F. (2003). Metal foams as compact high performance heat exchangers. *Mechanics of Materials*, 35(12), 1161–1176. https://doi.org/10.1016/S0167-6636(02)00226-8
[3] Calmidi, V. V., & Mahajan, R. L. (2000). Forced convection in high porosity metal foams. *Journal of Heat Transfer*, 122(3), 557–565. https://doi.org/10.1115/1.1285902
[4] Dukhan, N. (2012). *Metal foam heat exchangers and heat sinks*. Nova Science Publishers.
[5] Bhattacharya, A., & Mahajan, R. L. (2004). Metal foam heat exchangers. In *Thermal Management of Microelectronic Equipment* (pp. 375–386). ASME.
[6] Jiang, Y., Li, Z., & Liu, W. (2020). Numerical study on the thermal performance of metal foam-based heat sinks using CFD. *Applied Thermal Engineering*, 164, 114511. https://doi.org/10.1016/j.applthermaleng.2019.114511
[7] Zhao, C. Y. (2012). Review on thermal transport in high porosity cellular metal foams with open cells. *International Journal of Heat and Mass Transfer*, 55(13-14), 3618–3632. https://doi.org/10.1016/j.ijheatmasstransfer.2012.03.005
[8] Ferziger, J. H., & Perić, M. (2020). Computational Methods for Fluid Dynamics. Springer
[9] Versteeg, H. K., & Malalasekera, W. (2007). An Introduction to Computational Fluid Dynamics: The Finite Volume Method. Pearson Education.
[10] Pope, S. B. (2000). Turbulent Flows. Cambridge University Press.
[11] Moin, P., & Mahesh, K. (1998). "Direct Numerical Simulation: A Tool in Turbulence Research." Annual Review of Fluid Mechanics, 30(1), 539-578.
[12] Wilcox, D. C. (2006). Turbulence Modeling for CFD. DCW Industries.
[13] Spalart, P. R. (2009). "Detached-Eddy Simulation." Annual Review of Fluid Mechanics, 41, 181-202.
[14] Sagaut, P. (2006). Large Eddy Simulation for Incompressible Flows: An Introduction. Springer.
[15] Hetsroni, G., Gurevich, M., & Rozenblit, R. (2008). Forced convection heat transfer in a channel filled with porous medium: Metal foam. *International Journal of Heat and Mass Transfer*, 51(5-6), 958–970. https://doi.org/10.1016/j.ijheatmasstransfer.2007.04.045