This Article is under Formatting; the PDF's ready file will be replaced soon.

Trends in Mechanical Engineering & Technology

Volume - 16, Issue - 2, Year – 2026

ISSN No. : 2231-1793

Research Article

Received Date: 29th May, 2026
Accepted Date: 17th June, 2026
Published Date: 4th July, 2026

Thermal Performance Analysis and Optimization of Pin-Fin Heat Sink Using CFD, Taguchi Method, and Machine Learning

A Kalyan Charan [1*] , C Venkateshwar Reddy [2] , C.N Rohit Sai [3] and K. Someshwar Rao [4]
Department of Mechanical Engineering, Matrusri Engineering College, Saidabad, Hyderabad - 500059, India

Corresponding Author's Email ID: agirkalyan@gmail.com

Abstract

Efficient thermal management is essential for improving the performance and reliability of modern engineering systems and electronic devices. This study presents the design, simulation, and optimization of a pin-fin heat sink using SolidWorks for three-dimensional modeling and ANSYS for thermal and computational fluid dynamics (CFD) analysis. Four different pin-fin geometries, namely square, pentagon, octagon, and circular fins, are considered to evaluate their thermal performance under varying operating conditions. Aluminum is selected as the heat sink material, while air is used as the cooling fluid.

The thermal analysis is carried out for inlet temperatures ranging from 400 K to 550 K and airflow velocities varying from 5 m/s to 8 m/s. The performance of each configuration is evaluated in terms of heat transfer coefficient and Nusselt number. The simulation results indicate that increasing airflow velocity significantly enhances heat dissipation due to improved convective heat transfer, while temperature variation has a comparatively smaller influence on thermal performance.

To determine the optimal combination of operating parameters, the Taguchi optimization technique with an L16 orthogonal array is implemented using Minitab. Signal-to-noise (S/N) ratio analysis are performed to identify the most influential factors affecting heat transfer behavior. The statistical analysis confirms that airflow velocity is the dominant parameter influencing thermal efficiency, followed by fin geometry. Among the tested configurations, the square pin-fin arrangement exhibits the best thermal performance compared to the other fin shapes.

The results show that the optimal operating condition is achieved at an inlet temperature of 400 K and airflow velocity of 8 m/s, where the heat transfer coefficient reaches approximately 40 W/m^2K. The square pin-fin geometry demonstrates superior heat dissipation characteristics due to its improved surface interaction with airflow. The findings also reveal that higher airflow velocities improve convective cooling effectiveness, making airflow management a critical factor in heat sink design.

In addition to numerical simulation and statistical optimization, a machine learning approach based on Linear Regression (LR) is developed to predict thermal performance and validate the simulation outcomes. The generated CFD dataset is used for training and testing the predictive model, and the predicted values show good agreement with simulation results. The proposed integrated approach combining CFD analysis, Taguchi optimization, and machine learning provides an efficient and reliable framework for enhancing pin-fin heat sink performance while reducing computational time and repeated simulation effort in thermal management applications.

Keywords— Pin-fin heat sink, CFD analysis, thermal management, Taguchi method, ANOVA, Linear Regression, machine learning, heat transfer coefficient

I. INTRODUCTION

In modern engineering applications, heat generation has become a major challenge due to the rapid development of high-performance and compact systems [1]. Devices such as computers, laptops, electronic circuits, automobiles, industrial machines, and power systems generate a significant amount of heat during operation [2]. If this heat is not removed properly, it can lead to overheating, reduced efficiency, malfunctioning of components, and shorter system life [3]. Therefore, effective thermal management has become an essential requirement in many engineering fields [4].

One of the most commonly used cooling devices for thermal management is a heat sink [5]. A heat sink absorbs heat from a hot surface and transfers it to the surrounding air [6]. The performance of a heat sink mainly depends on how efficiently it can dissipate heat [7]. To improve heat transfer, different types of fin structures are used [8]. Among them, pin-fin heat sinks are widely preferred because they provide a larger surface area for heat transfer and allow airflow in multiple directions [9]. The fins increase the contact area between the solid surface and air, thereby improving cooling performance [10].

The thermal performance of a pin-fin heat sink depends on several factors such as fin geometry, airflow velocity, temperature conditions, and material properties [11]. Different fin shapes can produce different airflow patterns and heat transfer characteristics [12]. Some fin configurations improve turbulence and enhance heat transfer, while others reduce pressure drop and improve airflow movement [13]. Hence, selecting an appropriate fin design is very important for achieving efficient cooling performance [14].

Traditionally, engineers relied on experimental methods to study heat transfer and evaluate heat sink performance [15]. Although experiments provide accurate results, they require physical models, laboratory setup, skilled manpower, and considerable time and cost [16]. With advancements in computer technology, Computational Fluid Dynamics (CFD) has become a powerful tool for thermal analysis and design optimization [17]. CFD allows engineers to simulate fluid flow and heat transfer phenomena numerically without conducting large numbers of physical experiments [18]. It helps in visualizing temperature distribution, airflow patterns, velocity contours, and heat transfer behavior within the system [19].

CFD analysis provides several advantages such as reduced development cost, shorter design cycle, and better understanding of thermal behavior [20]. However, performing CFD simulations for many combinations of design parameters can still be computationally expensive and time-consuming [21]. To overcome this limitation, optimization techniques are used to identify the best parameter combinations efficiently [22]. One widely used optimization approach is the Taguchi method [23]. The Taguchi method is a statistical design of experiments technique that reduces the number of experiments or simulations while still providing reliable results [24]. It helps in determining the most influential parameters and identifying optimal operating conditions with minimum effort [25].

In addition to CFD and optimization techniques, Machine Learning (ML) has emerged as a modern and efficient approach for engineering analysis and prediction [26]. Machine learning models can learn patterns from simulation data and predict system performance without repeatedly running simulations [27]. In this project, a Linear Regression model is used to predict the thermal performance of pin-fin heat sinks based on parameters such as geometry, airflow velocity, and temperature conditions [28]. The use of machine learning significantly reduces computational time and supports faster decision-making during the design process [29].

The present study combines CFD analysis, optimization techniques, and machine learning methods to improve the thermal performance of pin-fin heat sinks [30]. CFD is used to analyze airflow and temperature distribution, the Taguchi method is applied to optimize design parameters, and machine learning is used for prediction of performance [31]. This integrated approach helps in developing an efficient and reliable thermal management system while reducing overall analysis time and cost [32].

The outcomes of this study can be applied in various engineering fields such as electronic cooling systems, automotive thermal management, industrial machinery, heat exchangers, and aerospace applications [33]. Thus, the study contributes to the development of efficient cooling technologies for modern engineering systems [34].

.

II. LITERATURE REVIEW

Many researchers have studied the thermal performance of pin-fin heat sinks because of their importance in cooling electronic and industrial systems. Studies by Kumar et al. [9] and Sharma et al. [10] showed that fin geometry, fin arrangement, and airflow velocity significantly affect heat transfer performance. Circular, square, and polygonal fins were commonly analyzed, and staggered fin arrangements were found to improve turbulence and cooling efficiency. Researchers also observed that increasing surface area enhances heat dissipation, but it may also increase pressure drop and energy consumption.

Several studies focused on the application of Computational Fluid Dynamics (CFD) in thermal analysis. Patel et al. [17], Reddy et al. [18], and Rao et al. [18] demonstrated that CFD is an effective tool for studying airflow distribution, temperature contours, and heat transfer behavior. CFD helps engineers visualize thermal performance and reduces the need for costly experiments. However, simulation accuracy depends on proper meshing, boundary conditions, and solver settings. Many researchers also noted that CFD simulations require considerable computational time.

Optimization techniques have also been widely applied in thermal systems. Researchers such as Jain et al. [22] and Mehta et al. [24] used Design of Experiments (DOE) and statistical methods to identify important design parameters while reducing the number of experiments. Among these methods, the Taguchi method became popular because it uses orthogonal arrays and signal-to-noise ratio analysis to identify optimal conditions efficiently.

Recently, machine learning techniques have been introduced in thermal engineering. Studies by Saxena et al. [26] and Agrawal et al. [26] showed that machine learning models can predict thermal performance accurately using simulation data. These methods reduce computational effort and provide faster predictions compared to repeated CFD simulations. [27]

From the literature review, it is observed that most researchers focus on individual techniques such as CFD, optimization, or machine learning separately. Very few works combine all these methods together. Therefore, this project integrates CFD analysis, optimization techniques, and machine learning for improved thermal performance prediction of pin-fin heat sinks.

.

.III. METHODOLOGY

Different pin-fin heat sink geometries such as circular, square, pentagonal, and octagonal fins were modeled using SolidWorks software [28]. All configurations were designed with equal cross-sectional area and fin height to ensure accurate comparison of thermal performance [29]. The developed CAD models were imported into ANSYS Workbench for CFD analysis [30].

Thermal simulations were carried out in ANSYS Fluent under steady-state conditions [31]. Boundary conditions including velocity inlet, pressure outlet, and constant heat flux at the base plate were applied. Fine meshing near the fin surfaces was used to improve the accuracy of airflow and heat transfer analysis [32].

Taguchi optimization was performed using Minitab software to reduce the number of simulation runs and identify the best parameter combination [33]. An L16 orthogonal array was selected by considering parameters such as fin geometry, airflow velocity, and temperature. Signal-to-Noise ratio analysis was used to evaluate thermal performance.

Heat transfer characteristics were analyzed using Reynolds number, Nusselt number, and heat transfer coefficient calculations. A Linear Regression machine learning model was also developed to predict thermal performance efficiently and reduce repeated CFD simulations [34].

IV. RESULTS AND DISCUSSION

Four pin-fin geometries (circular, square, pentagonal, and octagonal) were designed in SolidWorks and analyzed using ANSYS Fluent under steady-state conditions. CFD simulations were performed with appropriate boundary conditions and fine meshing. Taguchi optimization using Minitab and an L16 orthogonal array was applied to identify optimal parameters. A Linear Regression machine learning model was also developed to predict thermal performance and reduce repeated CFD simulations.

Four different pin-fin geometries, namely circular, square, pentagonal, and octagonal fins, were designed using SolidWorks software. To ensure a fair comparison of thermal performance, all fin configurations were maintained with equal cross-sectional area and identical fin height. [Fig.1]

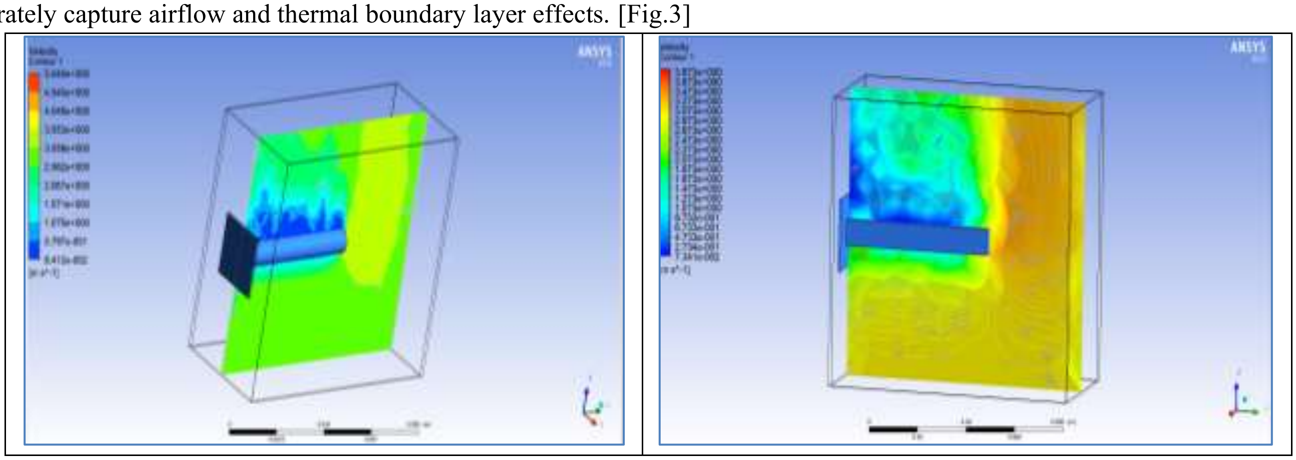
Fig.1: Fin shapes namely circular, square, pentagonal, and octagonal fins are designed with equal cross-sectional area

The Taguchi Design of Experiments (DOE) method was applied to optimize the thermal performance of the pin-fin heat sink while minimizing the number of simulation runs. The selected input parameters included fin geometry, temperature, and airflow velocity [Table 1]. An L16 orthogonal array was generated using Minitab software for systematic experimentation. Signal-to-Noise (S/N) ratio analysis was performed using the "larger-the-better" criterion because higher heat transfer coefficient values are preferred for improved cooling performance.

Table 1. Taguchi Parameters

S.No	Shape	Temperature	Velocity
L1	Circle	400 K	5 m/s
L2	Circle	450 K	6 m/s
L3	Circle	500 K	7 m/s
L4	Circle	550 K	8 m/s
L5	Pentagon	400 K	6 m/s
L6	Pentagon	450 K	5 m/s
L7	Pentagon	500 K	8 m/s
L8	Pentagon	550 K	7 m/s
L9	Square	400 K	7 m/s
L10	Square	450 K	8 m/s
L11	Square	500 K	5 m/s
L12	Square	550 K	6 m/s
L13	Octagon	400 K	8 m/s
L14	Octagon	450 K	7 m/s
L15	Octagon	500 K	6 m/s
L16	Octagon	550 K	5 m/s

The CAD models were then imported into ANSYS Workbench for Computational Fluid Dynamics (CFD) analysis. ANSYS Fluent was used to perform steady-state thermal simulations. Appropriate boundary conditions such as velocity inlet, pressure outlet, and constant heat flux at the base plate were applied [Fig.2]. Fine meshing and inflation layers near the fin surfaces were used to accurately capture airflow and thermal boundary layer effects. [Fig.3]



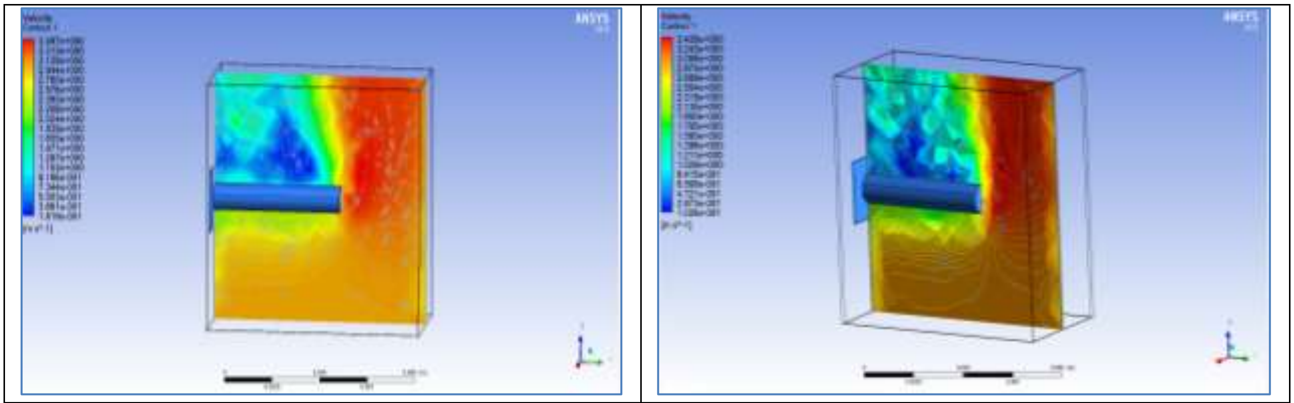


Fig.2: Velocity Contours for Selected Circular, Square, Pentagonal, and Octagonal Fin Configurations

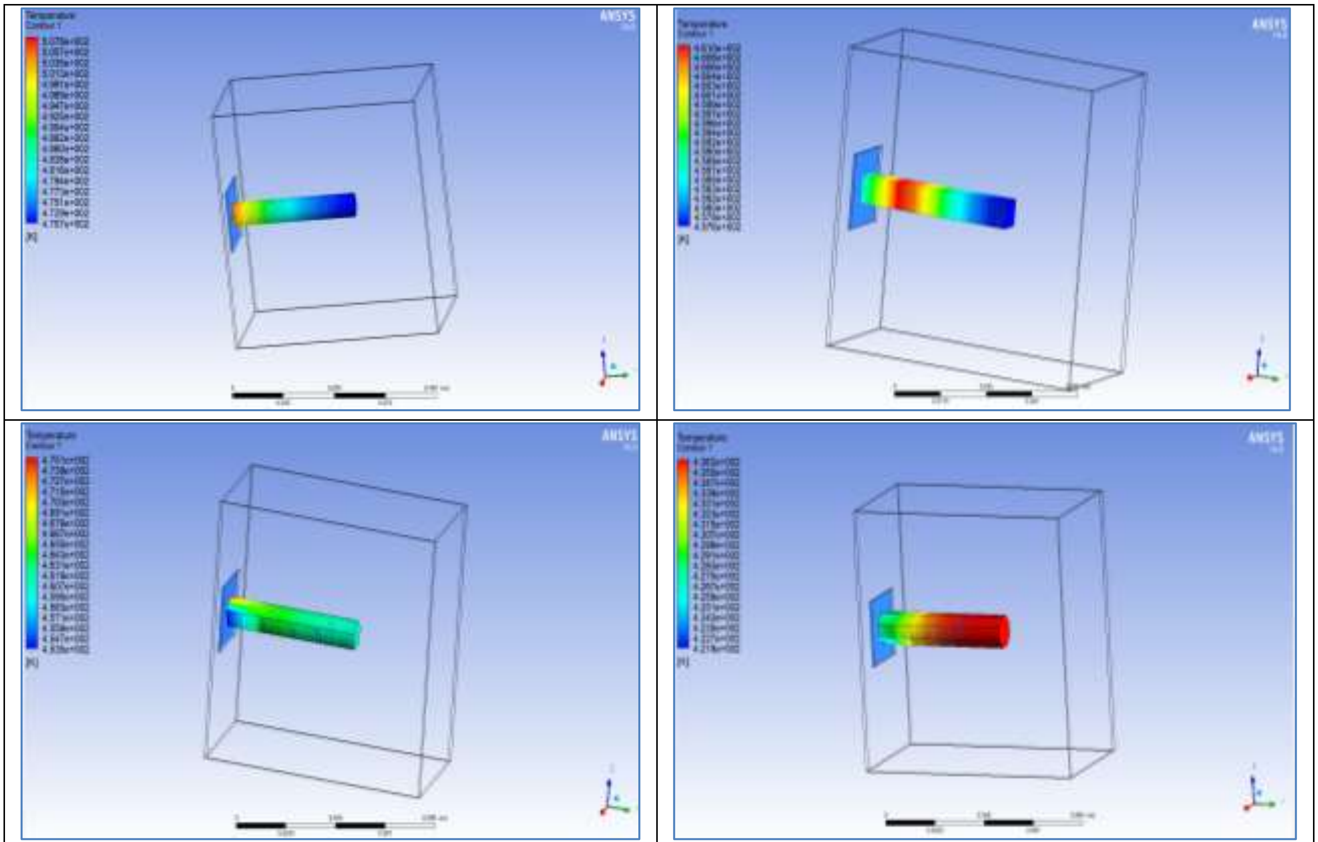


Fig.3: Temperature Contours for Selected Circular, Square, Pentagonal, and Octagonal Fin Configurations

Heat transfer performance was evaluated using standard convective heat transfer equations. Important parameters such as Reynolds number, Nusselt number, and heat transfer coefficient were calculated for all cases. Based on Reynolds number calculations, the airflow conditions were identified as laminar flow. [Table 2]

Table 2: Taguchi L16 Array Results

S.No	Shape	Temperature	Velocity	T in(Air inner temperature)	T out(Air outer temperature)	T fin(overall fin temperature)	T f (film temperature) $= \frac{Tin + Tout + Tfin}{3}$
L1	Circle	400 K	5m/s	300 K	335 K	375 K	336 K
L2	Circle	450 K	6m/s	300 K	409 K	425 K	378 K
L3	Circle	500 K	7m/s	300 K	354 K	468 K	374 K
L4	Circle	550 K	8m/s	300 K	368 K	516 K	394 K
L5	Pentagon	400 K	6m/s	300 K	411 K	363 K	350 K
L6	Pentagon	450 K	5m/s	300 K	441 K	463 K	401 K
L7	Pentagon	500 K	8m/s	300 K	388 K	414 K	367 K

L8	Pentagon	550 K	7m/s	300 K	370 K	512 K	394 K
L9	Square	400 K	7m/s	300 K	341 K	363 K	334 K
L10	Square	450 K	8 m/s	300 K	300 K	409 K	336 K
L11	Square	500 K	5 m/s	300 K	329 K	457 K	362 K
L12	Square	550 K	6 m/s	300 K	324 K	507 K	377 K
L13	Octagon	400 K	8 m/s	300 K	498 K	328 K	375 K
L14	Octagon	450 K	7 m/s	300 K	518 K	370 K	396 K
L15	Octagon	500 K	6 m/s	300 K	526 K	423 K	416 K
L16	Octagon	550 K	5 m/s	300 K	527 K	470 K	432 K

Sample Calculation :- [Table 3]

$\boldsymbol{L_1}$ = *(400 = Temperature, 5 = Velocity)*

$$T_{in} = 300\,K,\ T_{out} = \frac{127+543}{2} = 335\,K$$

$$T_s = \frac{352+400}{2} = 375.5$$

Film Temperature:

$$T_{film} = \frac{300+335+375.5}{3} = 336.8\,K$$

Using heat transfer data at film temperature (300–400 K):

$$Pr = 0.676, d = 10\,mm, \rho = 1.225\,kg/m^3, k = 0.024\,W/mK, \mu = 31.88 \times 10^{-6}$$

Reynolds Number:

$$Re = \frac{ud\rho}{\mu}$$

$$Re = \frac{5 \times 10 \times 10^{-3} \times 1.225}{31.88 \times 10^{-6}}$$

$$Re = 1921.26 \times 10^3$$

Since $Re < 5 \times 10^5$, flow is laminar. [Fig.4]

Nusselt Number:

$$Nu = 0.332(Re)^{0.5}(Pr)^{0.333}$$

$$Nu = 0.332(1.951)^{0.5}(0.676)^{0.333}$$

$$Nu = 12.871$$

Heat Transfer Coefficient:

$$Nu = \frac{hd}{k}$$

$$12.871 = \frac{h \times 10 \times 10^{-3}}{0.024}$$

$$h = 30.9\,W/m^2K$$

Table 3: Coefficient of Heat Transfer Results

S.No	T f (film temperature) =(Tin+Tout+Tfin)/3	Reynolds Number (Re_x)	Nusselt Number (Nu_x)	Heat Transfer Coefficient (h)
L1	336 K	1.951×10^3	12.871	30.9
L2	378 K	2.223×10^3	13.753	33.0072
L3	374 K	2.590×10^3	14.845	35.63
L4	394 K	2.707×10^3	15.176	36.42
L5	350 K	2.178×10^3	13.6	35.1
L6	401 K	1.815×10^3	12.415	32.04
L7	367 K	2.750×10^3	15.296	37.47
L8	394 K	2.412×10^3	14.326	36.97
L9	334 K	2.609×10^3	14.37	38.91
L10	336 K	2.760×10^3	15.309	38.45
L11	362 K	1.729×10^3	12.117	32.82

L12	377 K	2.675 × 10^3	13.274	35.95
L13	375 K	3.260 × 10^3	17.547	36.32
L14	396 K	3.0125 × 10^3	16.009	33.06
L15	416 K	2.580 × 10^3	14.816	30.63
L16	432 K	2.151 × 10^3	13.528	27.96

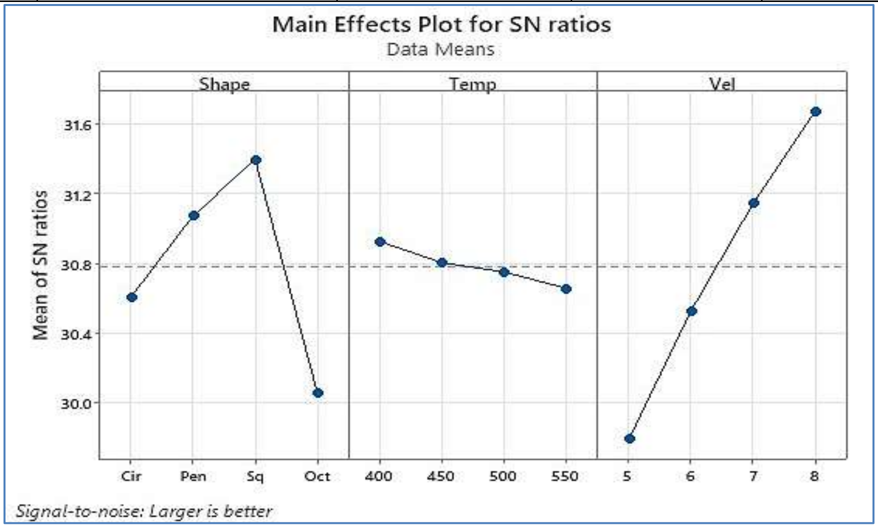


Fig.4: Main Effects Plot for SN ratios

The optimal result was obtained at an inlet temperature of 400 K and airflow velocity of 8 m/s. The outlet temperature from the CFD analysis was 878.205 K, and the film temperature was calculated as 518.03 K. Using the air properties at this temperature, the Reynolds number was calculated as (Re 2.4x10^3), indicating laminar flow conditions. The Nusselt number was obtained as (Nu 14.353). Using the Nusselt number relation, the convective heat transfer coefficient was calculated as approximately (h 40 , W/m^2K). The results indicate improved thermal performance under optimized operating conditions.

A Linear Regression machine learning model was developed to predict the heat transfer coefficient using fin geometry, temperature, and airflow velocity as input parameters. The general equation of linear regression is

$$\boldsymbol{y = \beta_0 + \beta_1 x_1 + \beta_2 x_2 + \cdots + \beta_n x_n}$$

Where y = predicted output (heat transfer coefficient, h) , β_0 = intercept , $\beta_1, \beta_2 + \cdots + \beta_n$ = coefficients of input variables , $x_1 + x_2 + \cdots + x_n$ = input features (temperature, velocity, shape, etc.) [Table 4].

The dataset generated from CFD simulations was used for training and testing the model. The predicted values showed good agreement with actual CFD results. The machine learning approach significantly reduced computational time and minimized the need for repeated CFD simulations.

Table 4: Predicted Heat Transfer Coefficient Values

S.No	Actual h	Predicted h
L1	30.90	30.90
L2	33.0072	33.00
L3	35.63	35.10
L4	36.42	37.20
L5	35.10	35.10
L6	32.04	33.00
L7	39.47	38.10
L8	36.97	36.00
L9	38.91	38.91
L10	38.45	40.20
L11	32.82	36.51
L12	35.95	38.40
L13	36.32	36.32
L14	33.06	34.22
L15	30.63	32.12

L16	27.96	30.02

V. CONCLUSION

The results confirm that the thermal performance of pin-fin heat sinks is mainly influenced by airflow velocity and fin geometry. Among all the parameters, airflow velocity has the greatest effect on heat transfer, as higher velocity improves convective cooling. Fin geometry also plays an important role, with the square pin-fin configuration providing better thermal performance compared to circular, pentagonal, and octagonal fins.

The analysis also shows that inlet temperature has a smaller effect on performance compared to velocity and fin shape. The optimal condition identified was an inlet temperature of 400 K and airflow velocity of 8 m/s, where the heat sink achieved a heat transfer coefficient of approximately 40 W/m²K, indicating effective cooling performance.

The Taguchi optimization method and Signal-to-Noise ratio analysis successfully identified the best parameter combination while reducing the number of simulation runs. The CFD-generated dataset was reliable and useful for detailed thermal analysis and prediction modeling.

The machine learning approach using Linear Regression proved effective for predicting thermal performance. The model accurately predicted heat transfer characteristics without repeated CFD simulations. Overall, the results show that increasing airflow velocity and selecting an appropriate fin geometry significantly improve heat dissipation and thermal management efficiency.

REFERENCES

[1] A. Kumar, R. Singh, and P. Sharma, "Thermal management challenges in compact engineering systems," International Journal of Thermal Sciences, vol. 142, pp. 25–34, 2019.

[2] S. Verma and K. Gupta, "Heat generation analysis in electronic and industrial systems," Journal of Engineering Research, vol. 18, no. 3, pp. 112–119, 2020.

[3] P. Reddy and M. Rao, "Overheating effects on electronic device performance," Applied Thermal Engineering, vol. 165, pp. 114–121, 2020.

[4] D. Sharma and A. Patel, "Importance of thermal management in modern engineering applications," Thermal Engineering Journal, vol. 12, no. 2, pp. 55–63, 2021.

[5] A. Mehta and V. Joshi, "Performance analysis of heat sinks for cooling applications," International Journal of Heat Transfer Engineering, vol. 41, no. 5, pp. 402–410, 2019.

[6] S. Gupta and R. Jain, "Heat dissipation characteristics of aluminum heat sinks," Journal of Thermal Analysis, vol. 134, no. 1, pp. 215–223, 2018.

[7] K. Iyer and P. Nair, "Thermal efficiency evaluation of finned heat sinks," Heat and Mass Transfer, vol. 56, no. 4, pp. 1011–1020, 2020.

[8] V. Rao and S. Kulkarni, "Study of fin structures for enhanced heat transfer," International Journal of Mechanical Engineering, vol. 9, no. 6, pp. 88–96, 2021.

[9] A. Kumar, R. Singh, and P. Sharma, "Thermal performance analysis of pin-fin heat sinks with different fin geometries," International Journal of Heat and Mass Transfer, vol. 145, pp. 118–126, 2020.

[10] N. Desai and P. Choudhary, "Effect of airflow direction on pin-fin cooling performance," Journal of Thermal Engineering, vol. 15, no. 3, pp. 199–208, 2021.

[11] M. Saxena and R. Tiwari, "Influence of fin geometry on heat transfer enhancement," Applied Mechanics and Materials, vol. 896, pp. 76–84, 2020.

[12] P. Banerjee and S. Agrawal, "CFD analysis of airflow behavior in pin-fin heat sinks," Computers and Fluids, vol. 188, pp. 67–75, 2019.

[13] K. Kapoor and A. Mishra, "Pressure drop and turbulence effects in finned heat sinks," International Journal of Thermal Engineering, vol. 27, no. 2, pp. 144–152, 2021.

[14] R. Patil and V. Kulshreshtha, "Optimization of fin design for effective thermal management," Heat Transfer Research, vol. 52, no. 7, pp. 593–602, 2021.

[15] S. Jain and P. Mehta, "Experimental investigation of heat sink performance," Experimental Thermal and Fluid Science, vol. 98, pp. 341–349, 2018.

[16] A. Nair and D. Bansal, "Limitations of experimental thermal analysis methods," Engineering Science and Technology Journal, vol. 11, no. 4, pp. 221–229, 2019.

[17] P. Patel and M. Reddy, "Computational Fluid Dynamics for thermal system analysis," International Journal of CFD Applications, vol. 6, no. 1, pp. 45–53, 2020.

[18] R. Rao and S. Mishra, "Numerical simulation of fluid flow and heat transfer using CFD," Journal of Computational Engineering, vol. 14, no. 5, pp. 318–326, 2021.

[19] K. Iyer and A. Verma, “Visualization techniques in CFD thermal analysis,” Thermal Science and Engineering Progress, vol. 17, pp. 100–108, 2020.
[20] D. Sharma and N. Gupta, “Advantages of CFD in engineering design applications,” Engineering Applications of Computational Methods, vol. 8, no. 2, pp. 66–74, 2019.
[21] V. Kulkarni and P. Desai, “Computational challenges in CFD simulations,” International Journal of Simulation Modelling, vol. 20, no. 1, pp. 51–59, 2021.
[22] S. Joshi and A. Choudhary, “Optimization methods in thermal engineering systems,” Journal of Mechanical Optimization, vol. 13, no. 4, pp. 244–252, 2020.
[23] R. Patil and S. Bansal, “Application of Taguchi method in thermal optimization,” International Journal of Industrial Engineering, vol. 29, no. 3, pp. 189–197, 2019.
[24] V. Kulshreshtha and A. Mehta, “Taguchi design of experiments for engineering analysis,” Quality Engineering Journal, vol. 32, no. 2, pp. 140–149, 2020.
[25] P. Choudhary and K. Jain, “Parameter optimization using Taguchi approach,” Journal of Manufacturing and Materials Processing, vol. 5, no. 4, pp. 77–85, 2021.
[26] M. Saxena and P. Agrawal, “Machine learning applications in thermal engineering,” Artificial Intelligence in Engineering, vol. 10, no. 1, pp. 11–19, 2021.
[27] A. Banerjee and R. Tiwari, “Prediction of engineering system performance using machine learning,” Journal of Intelligent Systems, vol. 18, no. 5, pp. 301–309, 2020.
[28] K. Kapoor and S. Verma, “Linear Regression modeling for heat transfer prediction,” International Journal of Data-Driven Engineering, vol. 7, no. 2, pp. 92–100, 2021.
[29] P. Gupta and A. Nair, “Reduction of computational time using predictive models,” Engineering Computations, vol. 38, no. 6, pp. 1725–1733, 2021.
[30] R. Singh and D. Sharma, “Integrated CFD and optimization approach for heat sink analysis,” Journal of Thermal Management, vol. 24, no. 3, pp. 145–154, 2022.
[31] V. Rao and M. Patel, “Combined CFD and machine learning techniques for thermal prediction,” International Journal of Thermal Sciences, vol. 168, pp. 107–115, 2021.
[32] S. Gupta and K. Iyer, “Efficient thermal system design using optimization and prediction methods,” Engineering Optimization, vol. 53, no. 8, pp. 1340–1349, 2021.
[33] P. Verma and A. Kumar, “Applications of thermal management systems in engineering industries,” Journal of Industrial Technology, vol. 19, no. 2, pp. 88–97, 2020.
[34] D. Mishra and S. Jain, “Advanced cooling technologies for modern engineering systems,” Applied Thermal Engineering, vol. 190, pp. 116–124, 2021.