International Journal of Trend in Scientific Research and Development (IJTSRD)
Volume 4 Issue 6, September-October 2020 Available Online: www.ijtsrd.com e-ISSN: 2456 – 6470

Thermal Performance of a Plate Fin Heat Sink with Half Round Pins using CFD

Dhiraj Kumar[1], Prof. Rohit Soni[2]

[1]Research Scholar, [2]Professor,

[1,2]Trinity Institute of Technology and Research, RGPV Bhopal, Madhya Pradesh, India

ABSTRACT

As electronic devices continue to advance towards high performance and miniaturization measurements, heat dissipation problems have become a major obstacle to their growth. Moreover, the conventional method of air-cooling did not fulfil the heat dissipation criterion of high density. Many scientists have researched the thermal efficiency of heat sinks, but the effects of half round pins on plate fin heat sinks have not yet been explored in the best possible way. In this work, a CFD model is developed to evaluate the effect of half round pin on the thermal performance of plate-fin heat sinks, compared with literary experimental data. Here in this work, a constant heat flux of 18750W/m2 along with a variable value of inlet air flow (i.e. 0.00092, 0.00218, 0.0033 and 0.00433 kg/s) has been applied. In particular, the plate-fin heat sink subject to impinging flow was compared to the conventional design (Plate-fin heat sink without half-round pins). Variation of conventional thermal resistance is compared with the approach proposed in the air mass flow rate of different values. The results of this study show that the base temperature along with the thermal resistance of the heat sink with half round pin is lower for the proposed design. Nusselt number in proposed approach is found to be 28.98 % higher than the conventional design at various values of the air mass flow rate due to increase in average heat transfer coefficient. Therefore, the developed approach has strong potential to be used to improve the thermal performance of heat sinks and hence to develop more advanced effective cooling technologies.

KEYWORDS: *Computational fluid dynamic (CFD), Heat sink, Half round pin, Thermal performance, Thermal resistance, Nusselt number*

How to cite this paper: Dhiraj Kumar | Prof. Rohit Soni "Thermal Performance of a Plate Fin Heat Sink with Half Round Pins using CFD" Published in International Journal of Trend in Scientific Research and Development (ijtsrd), ISSN: 2456-6470, Volume-4 | Issue-6, October 2020, pp.1387-1398, URL: www.ijtsrd.com/papers/ijtsrd33671.pdf

IJTSRD33671

I. INTRODUCTION

With the continuous development of electronic devices towards high performance and miniaturization size, heat dissipation problem has become a major obstacle to their development. Besides, the traditional air cooling method has been unable to meet the high-density heat dissipation requirement. Computer users prefer computers that having high-speed processors. The thermal design optimization of the heat sinks leads to minimize the size and weight of the heat sink, and then improve the heat removal in consequently increasing the speed of electronic devices. Electronic devices are increasingly miniaturized and the operating power of CPU increases. Besides, a larger amount of data processed by the CPU at a time causes greater heat generation. This development in the computer manufacturing makes the transfer of generated heat to the ambient becomes more difficult [1].

Generally, the heat generated by the processors is typically transferred to a heat sink (HS) by heat conduction, and then to the ambient by natural, mixed or forced convection. Low efficiency of heat removal of the heat sink possibly causing damage to the electronic component as the temperature rises [2]. This problem has motivated the computer manufacturers to employ sophisticated technology to improve the speed of electronic elements with increasing the heat removal. In contrary, the smallest size of the computers increases the overall flow resistance for the system and eventually suppresses the fluid flow between fins of the heat sink. This significantly influences the fan performance and affects its heat removal capability. Therefore, the heat sink must be designed properly to promote heat transfer and to avoid overheating of the electronic element.

1.1. Heat Sinks

A heat sink is a device that transfers the heat produced by a mechanical or electrical component to the surrounding medium, such as air or a liquid coolant. In other words, it keeps the component from overheating by absorbing its heat and dissipating it into the air. It accomplishes this task by maximizing the surface area in contact with the cooling medium surrounding it. This medium is often air, but it can also be a refrigerant, water, or oil.

Figure 1 Aluminium heat sink

1.1.1. Heat Sink Types

The heat sinks are classified into different categories based on different criteria. The major types, namely active heat sinks and passive heat sinks.

Active Heat Sinks

These are generally fan type and utilize power for cooling purpose. They can also be termed as Heat sink or fans. The fans are further classified as ball bearing type and sleeve bearing type. The ball bearing motor fans are preferred as their working span is longer and they are cheaper when it comes to a long span usage. The performance of these kinds of heat sink is excellent, but not for long term applications as they consist of moving parts and are a bit expensive as well.

Passive Heat Sinks

These do not possess any mechanical components and are made of aluminum finned radiators. These dissipate thermal energy or heat by using the convection process. These are most reliable than the active heat sinks; and, for efficient operation of passive heat sinks, it is recommended to maintain continuous air flow across their fins.

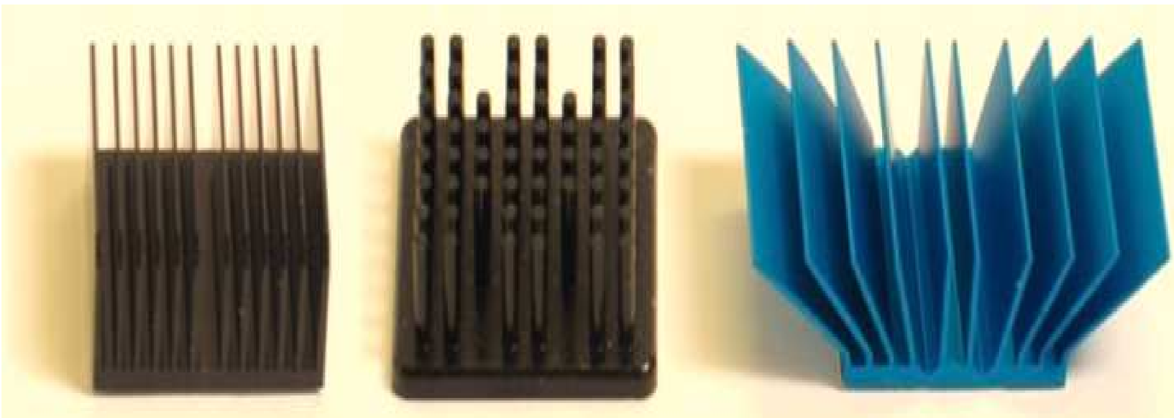


Figure 2 Based on fin arrangement i.e. straight, pin, and flared fins

1.1.2. Heat Transfer Principle of a Heat Sink

To understand the principle of a heat sink, consider ***Fourier's law of heat conduction***. Fourier's law of heat conduction, simplified to a one-dimensional form in the *x*-direction, shows that when there is a temperature gradient in a body, heat will be transferred from the higher temperature region to the lower temperature region. The rate at which heat is transferred by conduction, $\boldsymbol{q_x}$, is proportional to the product of the temperature gradient and the cross-sectional area through which heat is transferred.

$$q_x = -KA\frac{dT}{dx}$$

Now consider a heat sink in a duct, where air flows through the duct. It is assumed that the heat sink base is higher in temperature than the air.

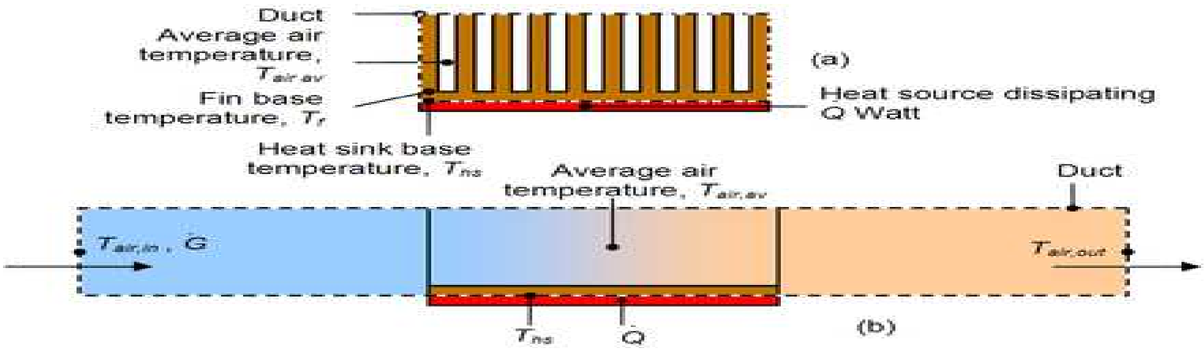


Figure 3 Heat Sink in a duct

Applying the conservation of energy, for steady-state conditions, and Newton's law of cooling to the temperature nodes shown in the diagram gives the following set of equations:

$$\dot{Q} = \dot{m}c_{p,in}(T_{air,out} - T_{air,in})$$

$$\dot{Q} = \frac{T_{hs} - T_{air,avg.}}{R_{hs}}$$

where, $T_{air,avg.} = \dfrac{T_{air,in} + T_{air,out}}{2}$

These above equations show that:-

A. When the air flow through the heat sink decreases, this results in an increase in the average air temperature. This in turn increases the heat sink base temperature. And additionally, the thermal resistance of the heat sink will also increase. The net result is a higher heat sink base temperature.
B. The increase in heat sink thermal resistance with decrease in flow rate will be shown later in this article.
C. The inlet air temperature relates strongly with the heat sink base temperature. For example, if there is recirculation of air in a product, the inlet air temperature is not the ambient air temperature. The inlet air temperature of the heat sink is therefore higher, which also results in a higher heat sink base temperature.
D. If there is no air flow around the heat sink, energy cannot be transferred.

1.1.3. Importance of Heat Sinks in Electronic Circuits

A heat sink is a passive heat exchanger, and it is designed to have large surface area in contact with the surrounding (cooling) medium like air. The components or electronic parts or devices which are insufficient to moderate their temperature, require heat sinks for cooling. Heat generated by every element or component of electronic circuit must be dissipated for improving its reliability and preventing the premature failure of the component.

- It maintains thermal stability in limits for every electrical and electronic component of any circuit or electronics parts of any system. The performance of the heat sink depends on the factors like the choice of a material, protrusion design, surface treatment and air velocity.
- The central processing units and graphic processors of a computer are also cooled by using the heat sinks. Heat sinks are also called as Heat spreaders, which are frequently used as covers on a computer's memory to dissipate its heat.
- If heat sinks are not provided for electronic circuits, then there will be a chance of failure of components such as transistors, voltage regulators, ICs, LEDs and power transistors. Even while soldering an electronic circuit, it is recommended to use heat sink to avoid over heating of the elements.
- Heat sinks not only provide heat dissipation, but also used for thermal energy management done by dissipating heat when heat is more. In case of low temperatures, heat sinks are intended to provide heat by releasing thermal energy for proper operation of the circuit.

1.1.4. Heat Sink Performance

The performance of heat sinks are a consequence of many parameters, including:

- Geometry
- Material
- Surface treatment
- Air velocity
- Interface with device

The effective thermal management of heat sinks is of priority concern of researchers. The lower thermal resistance, uniform temperature distribution, and lower maximum temperature on the base surface, lower pumping power, higher compactness and lower fabrication cost are still the essential requirements in heat sinks. Therefore the current study attempts to close this gap and discusses a CFD analysis to introduce new efficient cooling systems subject to impinging flow. The removed material from the fin base is attached to the plate fins in the form of half-round pins. In particular, a new thermal design for plate-fin heat sinks with half round pin will be developed to analyze the convective heat transfer in plate-fin heat sinks. Therefore, this research serves a platform to improve the thermal efficiency of plate-fin heat sinks through introducing new simple designs that offer easier fabrication and implementation compared to the aforementioned novel designs.

II. LITERATURE REVIEW

The performance of heat sinks has been the subject of numerical, analytical and experiments in many research projects in recent years. Most work has dealt with heat sinks in order to minimise the corresponding thermal resistance to improve electronic packaging heat transfer.

2.1. Previous work

A brief literature survey will be presented in this chapter to show how much has already been reported in the open literature on the heat sink.

Numerical studies were conducted by **Biber (1997)** to estimate a single isothermal channel pressure drop of impinging flow with variable inlet widths. Biber numerically developed the correlation for pressure drop coefficient of heat sinks with different heat sink parameters in this work. However, the correlation was not validated experimentally [1].

Duan and Muzychka (2007) studied the pressure drop of an impinging air cooled plate fin heat sink and presented a simple model and conducted experiments to validate the approximated effectiveness of the model [2].

In order to analyze the effect of fin shapes on the heat sink property, **Li et al. (2008)** investigated the distributions of the flow fields and temperature fields of heat sinks with short plate fins subjected to impinging flow. The pressure drop for three sets of fin shapes including rectangular, round-headed, and elliptic were studied in this paper. They found that the fin shapes have prominent influence on the secondary flow as well as flow separation [3].

Hung Yi Li and Chao (2009) measured performance of plate fin heat sink with cross flow cooling. Experimental results indicate that increase in Reynolds number decreases the thermal resistance. They observed that pressure drop increases with increase in Reynolds number, fin width and fin height. Study also reveals that increase in fin width degrades the thermal performance [4].

Li et al. (2009) experimentally and numerically investigated the thermal performance of plate fin heat sink subjected to impinging air flow with variable Reynolds numbers, the impinging distance, and the fin constructions [5].

Chang et al. (2009) investigated the air cooling module of electronic equipment and found the influences of heat load of heater and input current to cooler by experiments [6].

M.Dogan and M.Sivrioglu (2010) investigated mixed convection heat transfer from longitudinal fins inside horizontal channel. Wide range of modified Rayleigh number and fin heights and spacing's was used. They also investigated optimum fin spacing's to obtain maximum heat transfer. Air as a working fluid was used. Velocity of fluid was kept nearly constant ($0.15 \leq W_{in} \leq 0.16$ m/s) with a flow rate control valve so that Reynolds number was always about Re=1500.Results obtained from experiments show that optimum fin spacing's which gives maximum heat transfer is S=8-9mm and depends on value of Ra [7].

Wang et al. (2012) investigated the performance of a thermoelectric generator which has air cooling system and made two-stage optimized design [8].

Kumar and Bartaria (2013) assessed thermal performance and pressure drop of plate-elliptical pin fins heat sink(PEPFHS) which consists of some elliptical pins between those plate fins in-line arrangement. The range of parameters is regarded as air velocity is 6.5, 9.5 and 12.5 m/s. The length, height, thickness of this plate fin heat sink, and fin-to-fin distance are 51, 10, 1.5, and 5 mm, respectively; heating power is constant at 10 W in all cases. In addition, the number of elliptical pins is 9 pin fins, major and three different minor radiuses of elliptical pins are 5 mm, and 1.5,2, 2.5 mm, respectively. In order to solve the governing equations, the k-ε turbulent model is used by utilizing FLUENT 12.1 programme based on the finite volume method. The results indicate that thermal resistance and the Nusselt number of (PEPFHS) enhance with decreasing the minor radiuses of elliptical pins. Furthermore, the Nusselt number of (PEPFHS) is higher than that of the plate pin (PFHS). However, the pressure drop of (PFHS) is lower than that of the (PEPFHS) [9].

Abbas Jubear and Hamadani (2015) investigated the effect of fin height on thermal performance of plate fin array in natural convection. Study reveals that the heat transfer performance increases with fin height for constant fin spacing. They established correlation between Nusselt number and fin height by keeping all other geometrical parameters constant as for fin height varied between 10 to 45 mm [10].

Mao-Yu and Cheng (2015) conducted simulation studies by using COMSOL Multiphysics software to examine the thermal performance under forced convection for the heat sink designed in their previous research .Reynolds number (Re) range from 6468 to 45919 was studied and data obtained were compared with experimental data from other investigators. The results showed the highest heat transfer performance gained when a small hollow (D_h/D_b) < 0.15 is used in the base plate heat sink [11].

Saraireh (2016) made the three-dimensional computational fluid dynamics (CFD) simulation of fluid flow and heat transfer capabilities for the plate fin and pin fin heat sinks with different inlet velocities [12].

Ling et al. (2017) investigated the flow characteristics of non-Newtonian nanofluids flow in shell side of helical baffled heat transfer combined with elliptic tubes. They pointed out that a remarkable heat transfer enhancement can be obtained [13].

Maji et al. (2017) studied numerically the heat transfer through a pin fin with different numbers, shapes, and sizes of perforation under forced convection by using inline and staggered arrangements. All the perforated fin heat performance and pressure drop were compared with the corresponding solid fin under the same conditions. ANSYS 14 fluent software was used to design the system models. Heat flux of 5903 w/m^2 was applied at the bottom of the base plate which has an area of (0.1 × 0.1) m^2 and a thickness of 3mm, where the fins are mounted either in inline of staggered. The results showed that all perforated fins had higher thermal performance than the solid fins, especially with a staggered arrangement. The Nu number increases and the pressure drop decreases as the perforation number and size increase. The maximum heat transfer rate obtained by using elliptical fins with elliptical perforation is higher by 40.5 % than that of the solid circular fin [14].

Maiti and Prasad (2017) carried out a computational study on the heat transfer performance and the pressure drop in a fin heat sink under forced convection. Solid cylinder, slotted cylindrical, and kidney fin geometries. Reynolds number ranged from 2000 to 11000. The results obtained were validated with experimental results from previous work, and they found that the higher heat transfer rate acquired by using slotted kidney fin shapes with a staggered arrangement. Moreover, the decrease in pressure drop associated with the slotted fin was higher than that associated with the solid fin for both geometries, cylindrical and kidney [15].

Tang et al. (2017) made numerical simulations to analyze heat transfer characteristics of jet impingement with a novel single cone heat sink. It is found that the heat transfer capability of a cone heat sink with fluid impingement is better than that of a conventional plate fin heat sink [16].

Khattak and Ali (2019) made a critical review to summarize the development of air-cooled heat sink with forced flow arrangement [17].

Naphon et al. (2019) investigated the flow performance of micro-channel heat sink with impinging nanofluids flow employing experiments and numerical simulations [18].

Hussain et al. (2019) developed a CFD model to compare the flow and thermal characteristics for the plate fin heat sink of fillet profiles in parallel flow arrangement with that of a rectangular channel without fillet profiles in impinging flow. Investigation develops a computational fluid dynamics (CFD) model, validated through comparison with an experimental data from the literature, which demonstrates the effect of flow direction and fillet profile on the thermal performance of plate-fin heat sinks. In particular, a plate-fin heat sink with fillet profile subject to parallel flow has been compared with the conventional design (plate-fin heat sink without fillet profile subject to an impinging flow) and satisfactory results have been perceived. The results of the study shows that the base temperature along with the thermal resistance of the heat sink is lower for the proposed design [19].

2.2. Problem Formulation

The optimization of plate fin or other geometry of heat sink dimensions is still in primary research. This optimization should ensure low and uniform temperature distribution, as well as low pressure drops. Some of the researchers provided optimization of heat sink models, those model pave the way for upcoming optimization model that will guarantee the maximum performance of future PFHS.

Finally the growing interests in the PFHS, which is evident by the number of studies available, leads to conclusions that research in this area will give us optimized solution in this decade for cooling of electronic devices.

Despite the above improvement, different performance rates have been recorded in previous plays. For example, [19] says that the cooling efficiency of plate-fin heat sinks with fillet profiles is approximately 13 percent higher than that of sinks with conventional configuration. More effective designs are required to demonstrate the feasibility of enhancing the cooling performance of the platform heat sinks with a fillet profile.

2.3. Research objectives

In this analysis, the thermal characteristics of a plate fin heat sink with half round pin(which is removed from the material of the fin base and is attached to the plate fins) was investigated using a 3-dimensional numerical (3-D) simulation. The simulation programme ANSYS 17.0 was used for study of the heat transfer physiognomies of a plate fin heat sink with half round pin.

The main objectives of the present work are as follows:

- To analyze the thermal characteristics of plate fin heat sink with half round pin.
- To develop plate fin heat sink model and validation on CFD model will be carried out with comparison of previous model.
- Effect of half round pin in plate fin heat sink, thermal characteristics is analyzed by parameters such as the Nusselt number, Base temperature, and Thermal resistance.
- Calculating the effects of flow rate variation in the performance of the plate fin heat sink with half round pin.

III. METHODOLOGY

3.1. Computational Fluid Dynamics

Computational fluid dynamics is the computer based analysis by which we can analyses the various things like fluid flow, pressure distribution, heat transfer, and related to the phenomenon in the chemical reactions.

There are the three main elements for the processing of the CFD simulations: the pre-processor, solver, and post-processor are described.

Pre-processor: A pre-processor is defined to the geometry regarding the problem. And it is fixed into the domain for the computational analysis and then yields the mesh associated with geometry. Here also put the nomenclature like inlet, outlet, and wall etc. Usually, the finer the mesh associated with geometry into the CFD analysis offers more solution that is accurate. Fineness for the grid additionally determines the computer hardware and much more time needed for the calculations.

Solver: - The calculations is done by using the numerical solution methods in the solver processor. You will find the countless numerical practices that are utilized for the computations for example:-the finite factor method, finite amount technique, the finite huge difference technique additionally the spectral strategy. Most of them in CFD codes use finite volume method. The finite volume method is used in this project. The solver perform the steps: that is following

- Firstly the fluid movement equations are integrated on the control volumes (leading to the actual preservation of appropriate properties for every finite amount),
- Then these key equations tend to be discretised (creating algebraic equations through converting of the fundamental fluid movement equations),
- And then finally an iterative method is used to fix the algebraic equations.
- Pressure based paired option method CFD rule is used for re solving the simulations in this task.

Post-Processor: The post-processor is offered to the visualization for the total link between the solutions. It includes the ability to display the mesh and geometry also. As well as in this processor we could create the vectors, contours, and 2D and surface that is 3D of the issue solutions. Right Here the model can also be manipulated. In this method we could also look at cartoon of the problem.

3.2. Steps taken during the analysis

This chapter mentions the steps that have been taken place to achieve the objectives of the work.

- Firstly we design the CFD model of Plate Fin Heat Sink with half round pin fin on ANSYS 17.0 for CFD analysis.
- Meshing of model is done on CFD pre-processor.
- The boundary conditions are applied on the model and numerical solutions are calculated by using solver.
- The finite volume method is used in solving the problem.
- The solution is calculated by giving iterations to the mathematical and energy equations applied on model.
- Validation will be carried on CFD model with previous model.
- Applying formulas for calculating Base temperature, Nusselt number and Thermal resistance.
- The results can be visualized in the form contours and graphs by CFD post processor.
- Result analysis.

3.3. Calculation procedure

The ***average Nusselt number*** (Nū), estimates the output of a heat sink in the plate-fin and can be determined on the basis of the following equation:

$$\overline{Nu} = \frac{\bar{h} D_h}{K_a}$$

The latter is tabulated based on the ***mean temperature, T_m***, which is given by Equation:

$$T_m = \frac{T_{avg} + T_b}{2}$$

Where, T_b is base of fin temperature and T_{avg} represents average air temperature given by Equation:

$$T_{avg} = \left(\frac{T_{out} + T_{in}}{2}\right)_a$$

Where, T_{out} and T_{in} are outlet and inlet temperature of air respectively

The ***mean heat transfer coefficient, $\bar{h}$,*** is prescribed by:

$$\bar{h} = \frac{Q}{A_T (T_b - T_m)}$$

Where Q is the heat transfer rate per convection to the cold medium (air), A_T is the cumulative region subjected to the cooling fluid and T_m is the mean air temperature.

The ***heat transfer rate, Q,*** and ***total cooling area, A_T,*** can be expressed by Equations:

$$Q = \dot{m}_a c_{p,a} (T_{out} - T_{in})$$

Where, $\dot{m}_a$ and $c_{p,a}$ are the mass flow rate and the specific heat of air respectively

$$A_T = WL + 2N_f H(L + t) + 2B(L + W)$$

Where W and L are the heat sink width and length respectively, N_f describes the number of fins, H, t and B are the height, fins thickness, and base height respectively.

Substituting above two Eqs into Equation, $\bar{h} = \frac{Q}{A_T (T_b - T_m)}$, The mean heat transfer coefficient, $\bar{h}$, can be calculated:

$$\bar{h} = \frac{\dot{m}_a c_{p,a} (T_{out} - T_{in})}{[WL + 2N_f H(L + t) + 2B(L + W)](T_b - T_m)}$$

$$\therefore \overline{Nu} = \frac{\bar{h}D_h}{K_a} = \frac{\dot{m}_a c_{p,a}(T_{out}-T_{in})}{[WL+2N_f H(L+t)+2B(L+W)](T_b-T_m)} \times \frac{D_h}{K_a}$$

The ***pressure drop, ΔP***, between the inlet and outlet along with the ***thermal performance, R_{th}*** of plate-fin heat sinks is given by:

$$\Delta P = P_{in} - P_{out}$$

Where, P_{in} and P_{out} are inlet and outlet pressure respectively.

$$R_{th} = \frac{1}{\bar{h}.A_T}$$

IV. COMPUTATIONAL MODEL AND NUMERICAL SIMULATION

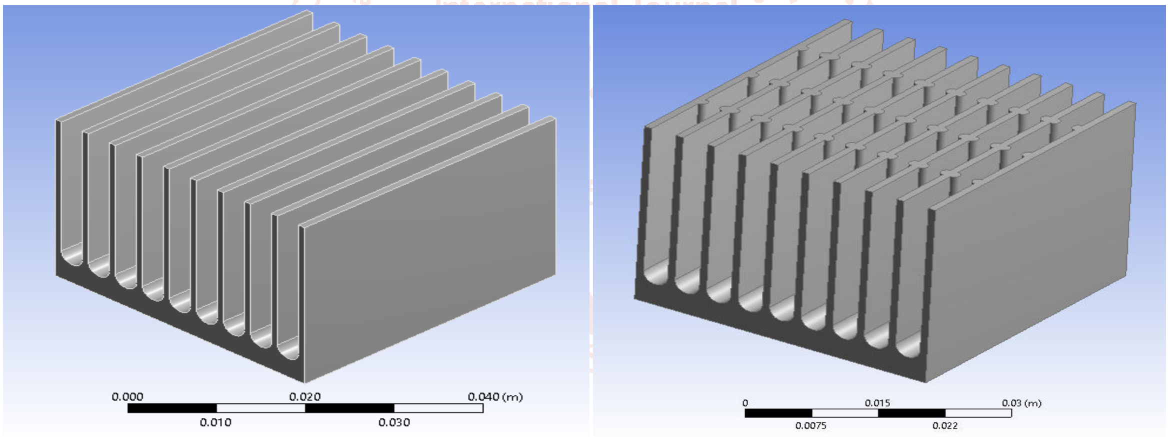
The study uses the CFD model in this section to investigate the heat transfer characteristics of the plate fin heat sink with half round pin fin. CFD review involves three major steps: (a) pre-processing, (b) solver execution, and (c) post-processing. The first step includes the creation of the geometry and mesh generation of the desired model, while the results are seen as expected in the last step. In the execution of the solver (medium) stage, the boundary conditions are fed into the model.

4.1. Geometrical specification of Plate fin heat sink

The geometry of plate fin heat sink performing the simulation study is taken from the one of the research scholar's **Hussain et al. (2019)** [19]with exact dimensions. After than in the proposed designs, the removed material from the fin base is attached to the plate fins in the form of half-round pins.

Table 1 Geometrical specification of plate fin heat sink

Parameters	Value
Base	40 mm* 39.77 mm
Base Thickness	5 mm
Channel width	3.3 mm
Channel thickness	1 mm
Fin height	28.6 mm
Fillet radius	1.5 mm



(a) **(b)**

Figure 4The geometrical model (a) Conventional design and (b) Proposed design

In the ANSYS FLUENT R17.0 pre-processor stage, a three-dimensional discreet plate fin heat sink model with half round pin was developed. While grid types are related to simulation output, the entire system is divided into the finite volume of Quad core tetrahedral grids in order to reliably measure the thermal characteristics of the plate fin heat sink with half round pin using the correct grids.

Table 2 Mesh details

The applied design	Number of elements
Plate-fin heat sinks (Conventional design)	1172636
Plate fin heat sink with half-round pins (Proposed design)	1225982

4.2. Model Selection and Solution Methods

Fluent 17.0 has been used to calculate computationally. In science, the method used to separate the governing equations was a finite element. The researchers used a simplified algorithm for this convective term, for the pressure-velocity coupling, the SIMPLE (Semi Implicit Method for Pressure Linked Equation) algorithm is applied.

A regular viscous-laminar equation with flow and energy equations was used to solve computationally

By solving guided equations i.e. the convective thermal transfer features can be accomplished. Navier-Stokes, continuity and energy calculations for energy conservation, mass conservation and the continuous movement of heat respectively. The below are the following assumptions:

1. The movement of the fluid is steady and incompressible.
2. By fact, the movement through the heat sinks is turbulent.
3. Fluid-solid conjugate is a three-dimensional conjugate.
4. All air physical properties (as coolant) are dependent on their ambient temperature.

The three dimensional Navier-Stokes equations are given as set of Equations:

$$\nabla.\left(\rho\,\vec{U}\,u\right) = -\frac{\partial p}{\partial x} + \frac{\partial \tau_{xx}}{\partial x} + \frac{\partial \tau_{yx}}{\partial y} + \frac{\partial \tau_{zx}}{\partial z}$$

$$\nabla.\left(\rho\,\vec{U}\,v\right) = -\frac{\partial p}{\partial y} + \frac{\partial \tau_{xy}}{\partial x} + \frac{\partial \tau_{yy}}{\partial y} + \frac{\partial \tau_{zy}}{\partial z}$$

$$\nabla.\left(\rho\,\vec{U}\,w\right) = -\frac{\partial p}{\partial z} + \frac{\partial \tau_{xz}}{\partial x} + \frac{\partial \tau_{yz}}{\partial y} + \frac{\partial \tau_{zz}}{\partial z}$$

Where$\vec{\boldsymbol{U}}$ is the fluid velocity with the components of u, v and w in x, y, and z direction respectively, $\boldsymbol{\rho}$ is the density of fluid, $\boldsymbol{p}$ is pressure and $\boldsymbol{\tau}$ is the tensor of viscous stress.

The energy equation is given by:

$$\nabla.\left(\rho\,\vec{U}\,h\right) = -p\nabla.\vec{U} + \nabla(k\nabla T) + \varphi + s_h$$

Where, h, k, T, φ and s_h are respectively aggregate enthalpy, thermal conductivity, temperature, form of dissipation and form of source.

The conductive energy equation (solid), which occurs through the various materials, is given by:

$$K_m\left(\frac{\partial^2 T}{\partial x^2} + \frac{\partial^2 T}{\partial y^2} + \frac{\partial^2 T}{\partial z^2}\right) = 0$$

$K_m$is the material's thermal conductivity.

Finally, the continuity equation is described by:

$$\nabla.\left(\rho\,\vec{U}\right) = 0$$

4.3. Thermophysical properties

For the present work aluminum alloy 6061 used as a material of plate-fin heat sink.

Table 3 Thermo-physical Properties of Aluminium alloy 6061

	Density (Kg/m³)	Specific Heat (J/Kg-K)	Thermal conductivity (W/m-K)
Aluminium alloy 6061	2700	896	167

Table.4. Thermo-physical Properties of Air

	Density (Kg/m³)	Specific Heat (J/Kg-K)	Thermal conductivity (W/m-K)
Air	1.225	1006.43	0.0242

4.4. Boundary Conditions

Here in this work, a constant heat flux of 18750W/m² along with a variable value of inlet air flow (i.e. 0.00092, 0.00218, 0.0033 and 0.00433 kg/s) has been applied as mention in base paper **Hussain et al. (2019)** [19]**.** Plate-fin heat sink (made of aluminium alloy 6061) is positioned in the wind tunnel (made of acrylic) and air impinges into the tunnel from a pressure tank through a mass flow meter. Electrical heaters warm the heat sink up and the temperature distribution at the base of the heat sink at different flow rates is measured computationally.

V. RESULTS AND DISCUSSIONS

This portion is intended to test the thermal performance of the plate fin heat sink with half round pin. Variations in the Base temperature, Nusselt number, and the Thermal resistance are calculated at various mass flow rate in order to investigate the performance of plate fin heat sink.

5.1. Validation of numerical computations

To validate the accuracy of developed numerical approach, comparison was made with the work reported in **Hussain et al. (2019)** [19]**.** The plate fin heat sink geometry that used for validation of numerical computations was considered to be as same as the geometry shown in Fig. 4 (a).

By way of CFD analysis, the value of the Base temperature has been measured at a different mass flow rate on the basis of which it determines the value of the amount of Nusselt and the thermal resistance. In contrast with values derived from analyses by **Hussain et al. (2019)** [19] the values in the Base Temperature, Nusselt numbers and thermal resistance estimates of the CFD modelling were compared [18].

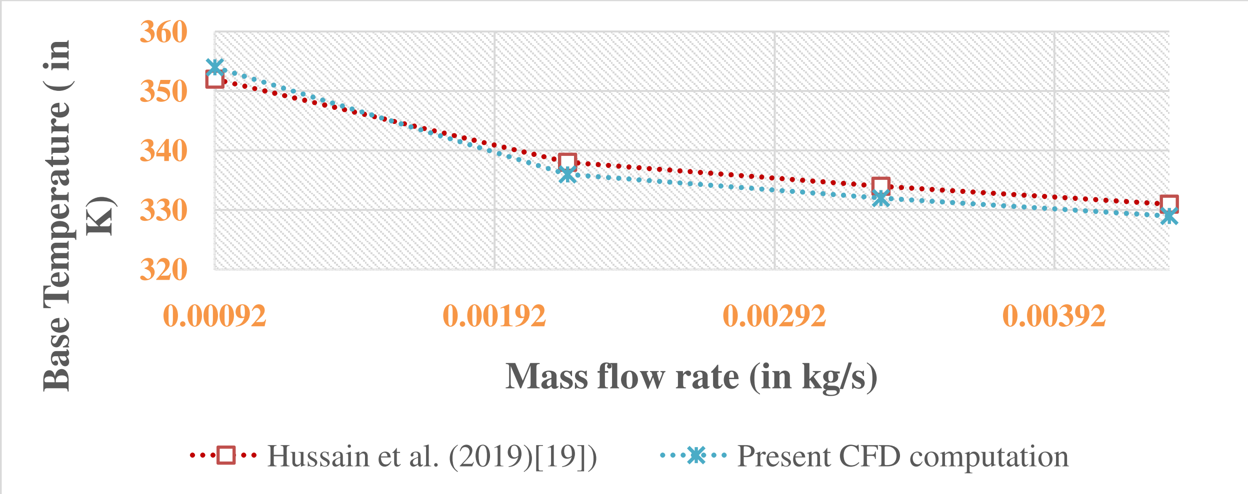


Figure 5 Base temperature values of Plate-fin heat sinks determined from CFD models opposed to the values derived from Hussain et al. (2019)[19]

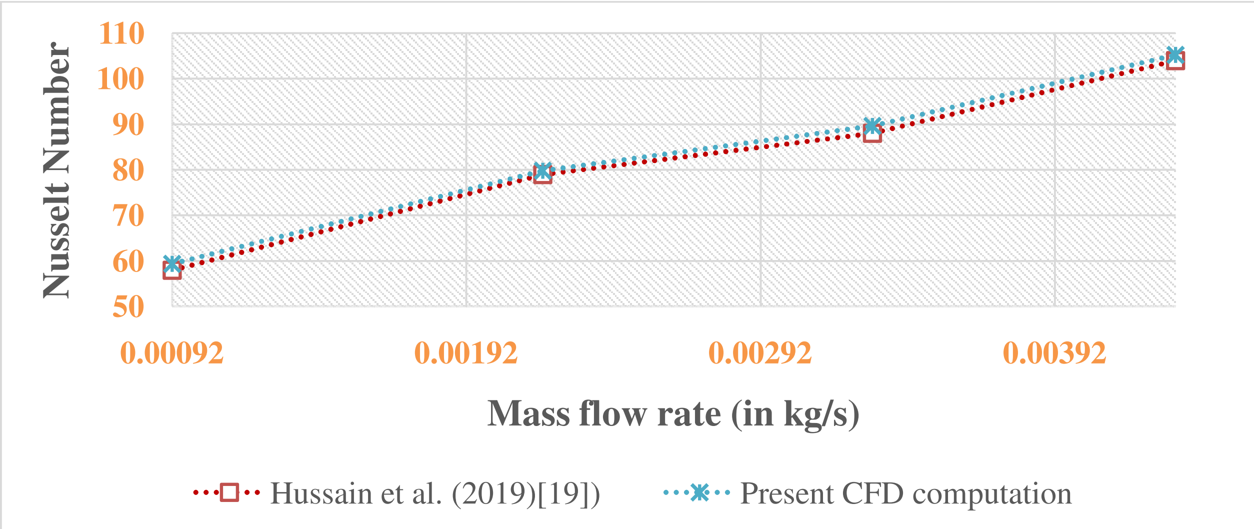


Figure 6 Nusselt number values of Plate-fin heat sinks determined from CFD models opposed to the values derived from Hussain et al. (2019)[19]

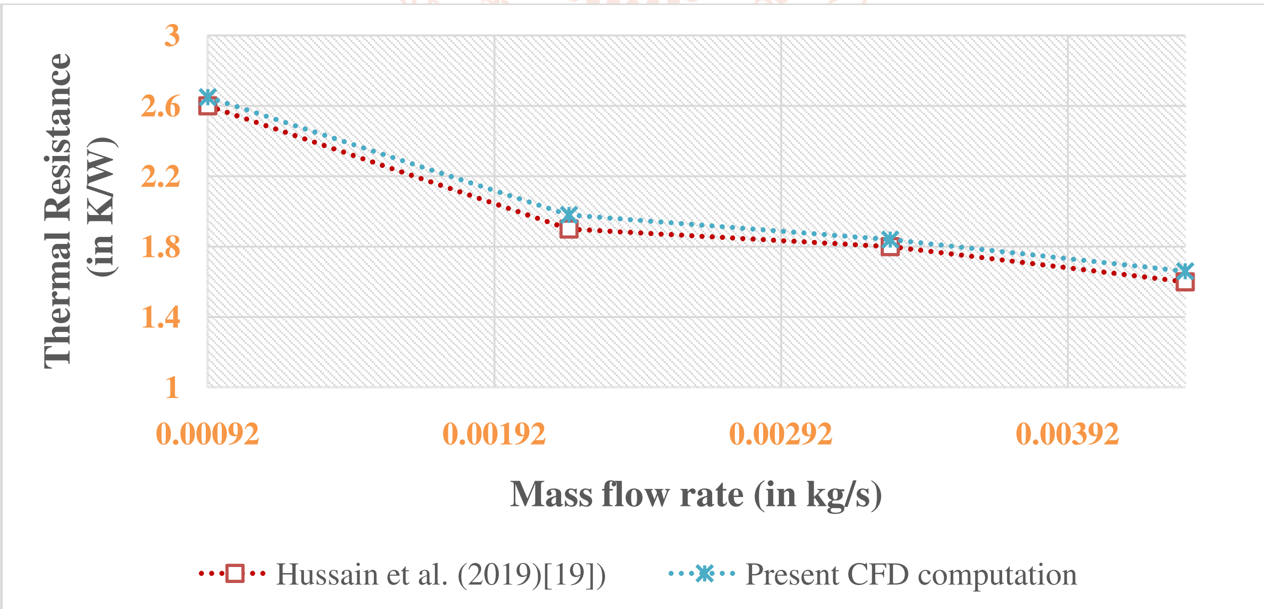


Figure 7 Thermal resistance values of Plate-fin heat sinks determined from CFD models opposed to the values derived from Hussain et al. (2019)[19]

From the above graph it is found that the value of Base temperature, Nu number, and thermal resistance calculated from numerical analysis is closer to value of Base temperature, Nu number, and thermal resistance obtained from the base paper which means that numerical model of plate fin heat sink is correct. There is much lesser difference between experimental and numerical values.

5.2. Effect of half round pin in the plate fin heat sink

From the numerical results and experimental data it is seen that variation tendencies in the values of base temperature, Nu number, and thermal resistance are qualitatively consistent. So, to analyzing the effect of half round pin in the plate fin heat sink to enhance heat dissipation, we take four mass flow rate of inlet air i.e. 0.00092, 0.00218, 0.0033 and 0.00433 kg/s. The boundary conditions were same as considered during the analysis of plate fin heat sink.

- **At mass flow rate of inlet air 0.00092 kg/s with a constant heat flux of 18750W/m²**

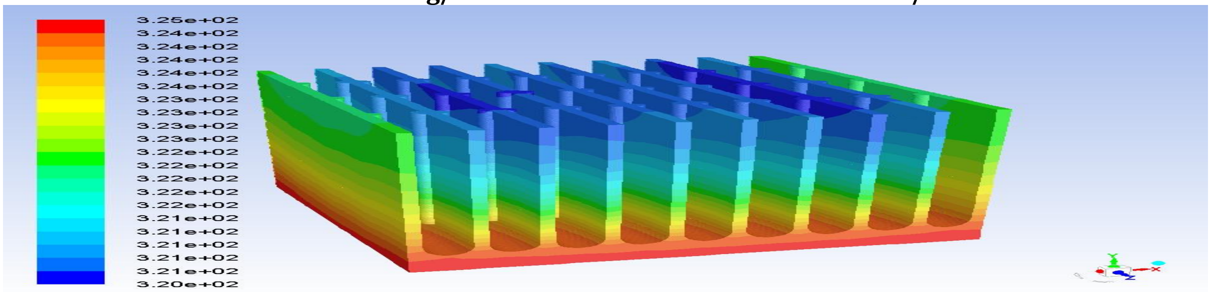


Figure 8 Temperature contour of plate fin heat sink with half round pin at a mass flow rate of 0.00092 kg/s

- **At mass flow rate of inlet air 0.00218 kg/s with a constant heat flux of 18750W/m²**

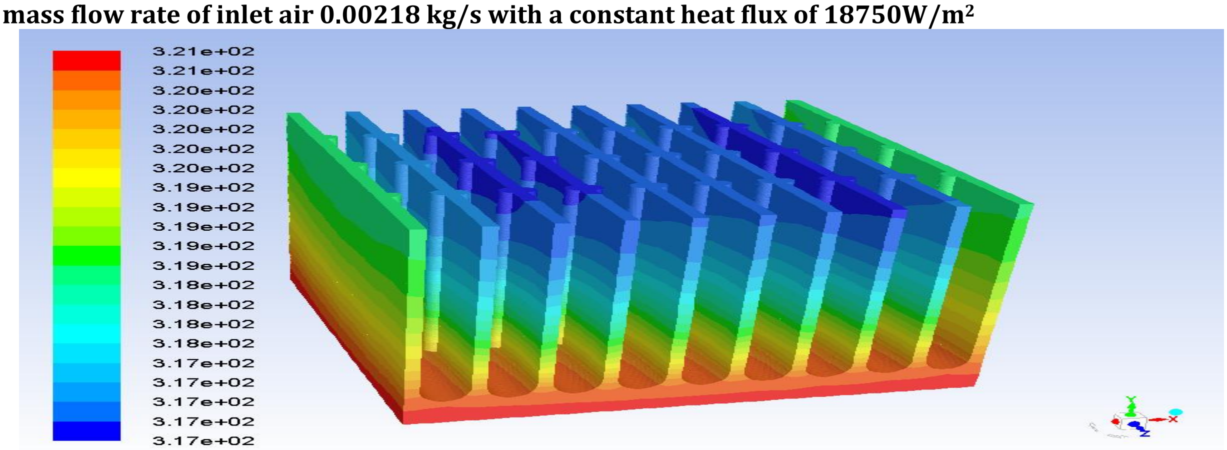


Figure 9 Temperature contour of plate fin heat sink with half round pin at a mass flow rate of 0.00218 kg/s

- **At mass flow rate of inlet air 0.0033 kg/s with a constant heat flux of 18750W/m²**

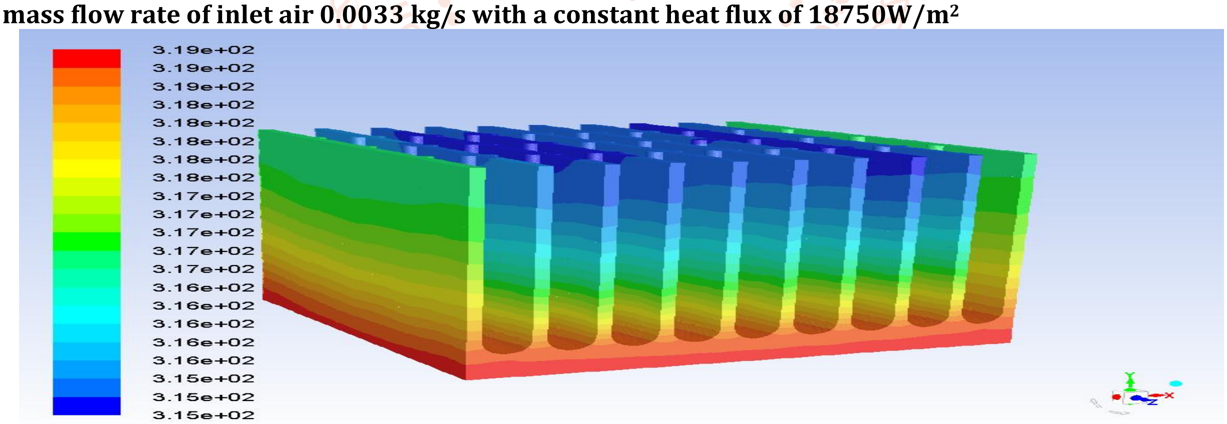


Figure 10 Temperature contour of plate fin heat sink with half round pin at a mass flow rate of 0.0033 kg/s

- **At mass flow rate of inlet air 0.00433 kg/s with a constant heat flux of 18750W/m²**

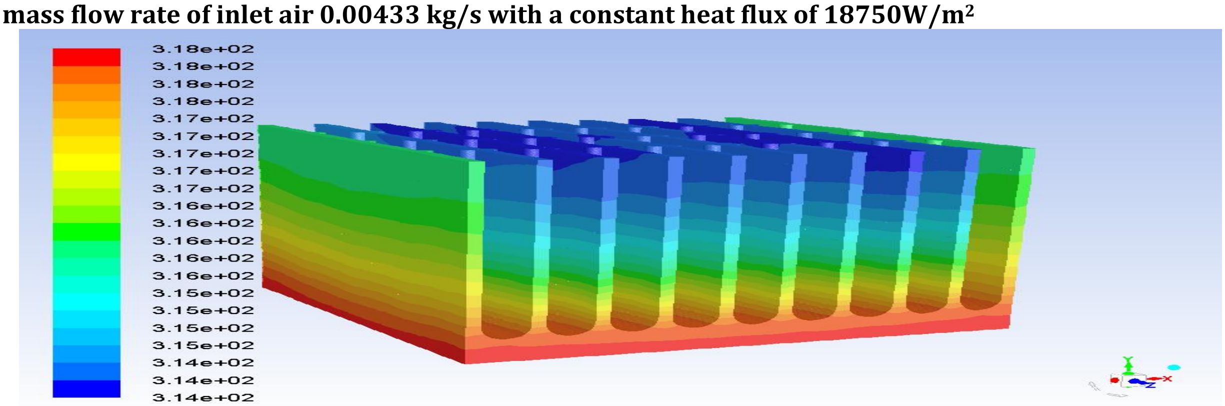


Figure 11 Temperature contour of plate fin heat sink with half round pin at a mass flow rate of 0.00433 kg/s

5.3. Comparison between Plate fin heat sink with and without half round pin

To improve previous understandings and to distinct the contribution of adding a half round pin to the overall thermal enhancement of proposed design, plate-fin heat sinks with and without half round pin are compared in this section.

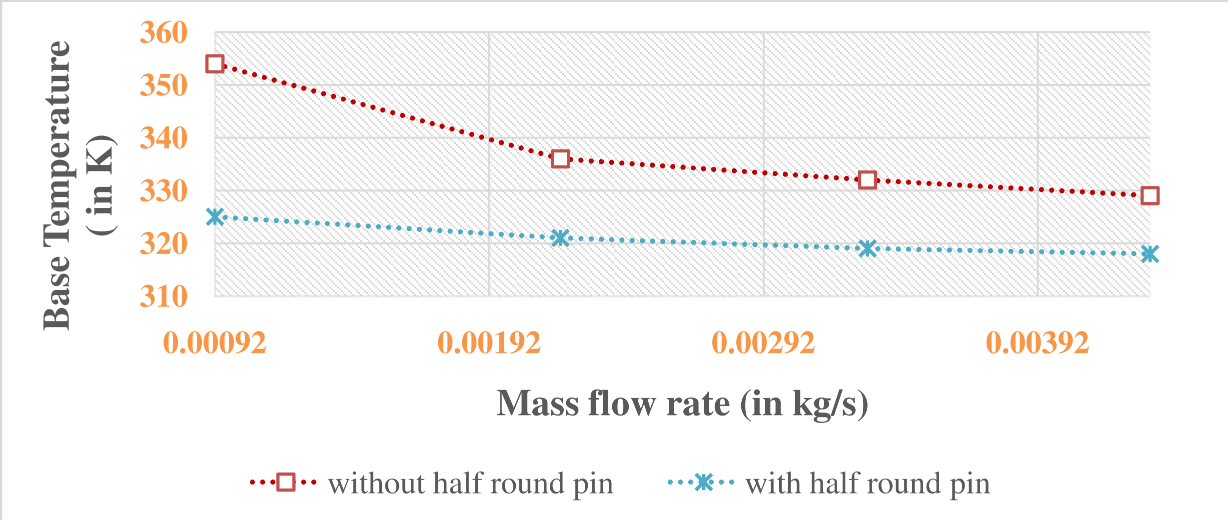


Figure 12 Base temperature values of Plate-fin heat sinks with and without half round pin

As it is expected in plate-fin heat sinks with half round pin, due to the increase in heat transfer area, more amount of heat is removed from the hot region. Therefore, the base temperature for plate-fin heat sinks with half round pin is less than those without half round pin. Moreover, the heat distribution is altered by the half round pin that proves the superior thermal performance of this configuration.

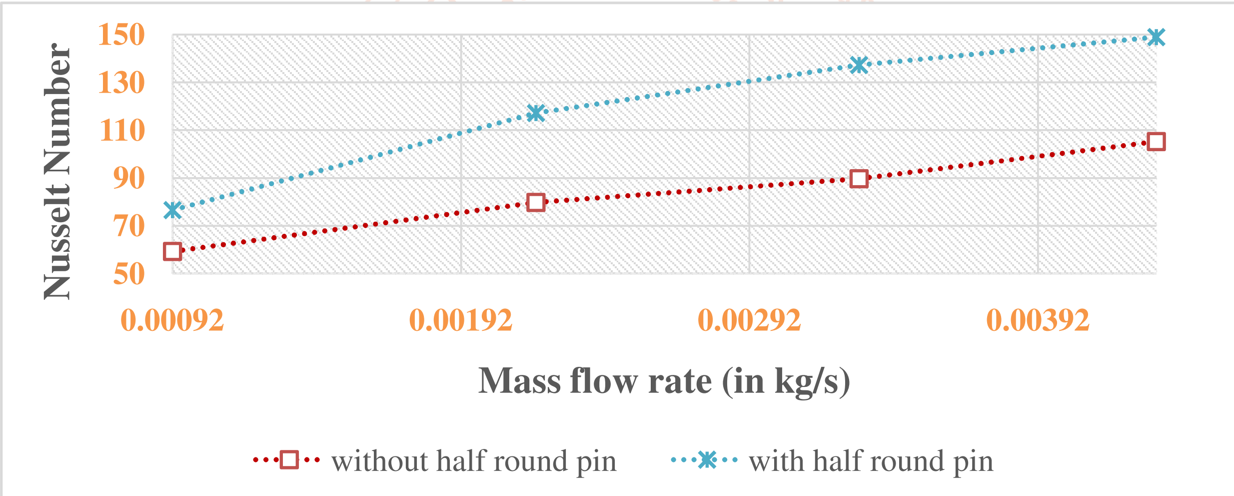


Figure 13 Nusselt number values of Plate-fin heat sinks with and without half round pin

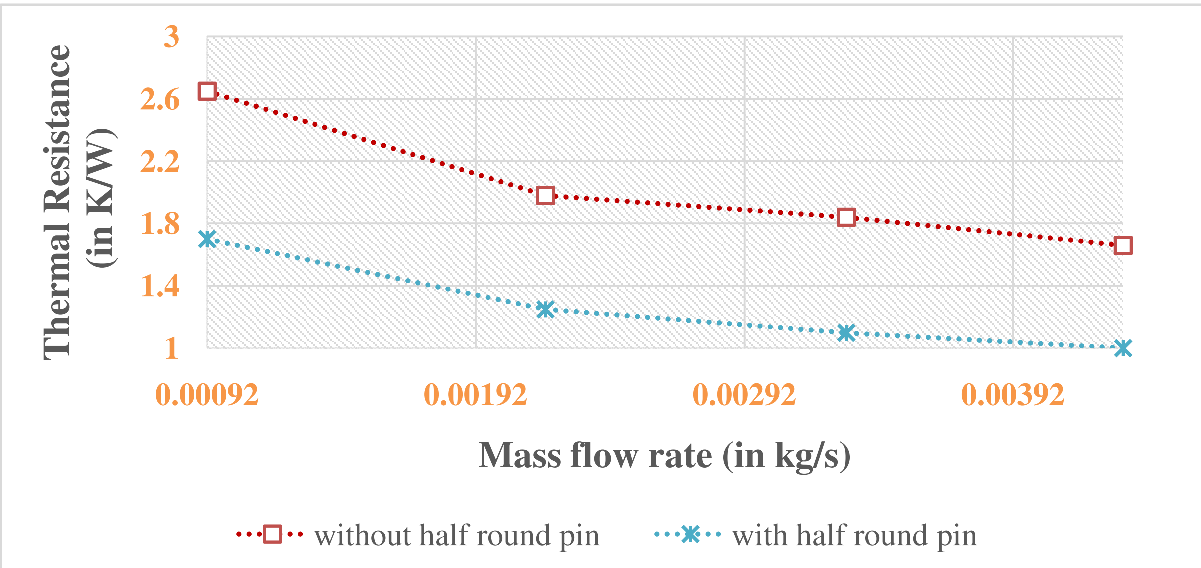


Figure 14 Thermal resistance values of Plate-fin heat sinks with and without half round pin

VI. CONCLUSIONS

This CFD research explores the impact of using half round pin on the performance of plate fin heat sink. A constant heat flux of 18750W/m^2 along with a variable value of inlet air flow (i.e. 0.00092, 0.00218, 0.0033 and 0.00433 kg/s) has been applied.A strong agreement has been seen in the comparison of the findings of this research with the existing experimental results of the literature. The effect of half round pin were measured and observed to influence the heat transfer and flow in a plate fin heat sink.The following conclusions can be drawn based on the provided results:

- Plate-fin heat sinks with half round pin, due to the increase in heat transfer area, more amount of heat is removed from the hot region. Therefore, the base temperature for plate-fin heat sinks with half round pin is less than those without half round pin. Moreover, the heat distribution is altered by the half round pin that proves the superior thermal performance of this configuration.
- Variation of thermal resistance for the conventional design is compared with the proposed approach at various values of the air mass flow rate. A notable difference (approximately 35 %) is evident between thermal resistance of the proposed design and conventional design that may prove the efficiency of proposed approach.
- A plausible explanation for such improvement in proposed approach is that, in addition to the effect of half round pin, the average heat transfer coefficient goes up due to the temperature gradient increase between the base of heat sink and the airflow. Therefore, the thermal resistance is lower as it is inversely proportional to the surface area and coefficient of heat transfer.
- The Nusselt number in proposed approach is found to be 28.98 % higher than the conventional design at various values of the air mass flow rate due to increase in average heat transfer coefficient.

REFERENCES

[1] Biber, C. R. Pressure drop and heat transfer in an isothermal channel with impinging flow. IEEE Trans. Compon. Packag. Manuf. Technol. Part A 1997, 20, 458–462.

[2] Duan, Z. P.; Muzychka, Y.S. Pressure drop of impingement air cooled plate fin heat sinks. Trans. ASME J. Electron. Packag. 2007, 129, 190–194.

[3] Li, X. C. Conjugate heat transfer of jet impingement on short fins with different shapes. In Proceedings of the 11th Intersociety Conference on Thermal and Thermo mechanical Phenomena in Electronic Systems, Orlando, FL, USA, 28–31 May 2008; pp. 49–56.

[4] Hung Yi Li and Chao, “Measurement of performance of plate-fin heat sinks with cross flow cooling”, International Journal of Heat and Mass Transfer, 2009.

[5] Li, H.Y.; Chen, K.Y.; Chiang, M.H. Thermal-fluid characteristics of plate-fin heat sinks cooled by impingement jet. Energy Convers. Manag. 2009, 50, 2738–2746.

[6] Chang, Y. W.; Chang, C. C.; Ke, M. T. Thermoelectric air-cooling module for electronic devices. Appl. Therm. Eng. 2009, 29, 2731–2737.

[7] M. Dogan, M. Sivrioglu, Experimental investigation of mixed convection heat transfer from longitudinal fins in a horizontal rectangular channel, International Journal of Heat and Mass Transfer 53 (2010) 2149–2158

[8] Wang, C. C.; Hung, C. I.; Chen, W. H. Design of heat sink for improving the performance of thermoelectric generator using two-stage optimization. Energy 2012, 39, 236–245.

[9] V. Kumar, D. V Bhartaria, CFD analysis of an elliptical pin fin heat sink using Fluent Ansys V 12.1, Int. J. Mod. Eng. Res. (3) 2 (2013) 1115-1122.

[10] Abbas Jubear and Hamadani, “The effect of fin height on free convection heat transfer from rectangular fin array”, International Journal of Recent Scientific Research Vol. 6, Issue 7, pp. 5318-5323, July – 2015

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

[12] Saraireh, M. Computational fluid dynamics simulation of plate fin and circular pin fin heat sinks. Jordan J. Mech. Ind. Eng. 2016, 10, 99–104.

[13] Ling, Z. Y.; He, Z. B.; Xu, T.; Fang, X. M.; Gao, X. N.; Zhang, Z. G. Experimental and numerical investigation on non-Newtonian nanofluids flowing in shell side of helical baffled heat exchanger combined with elliptic tubes. Appl. Sci. 2017, 7, 48

[14] Maji A., Bhanja D. , Patowari P. K., Numerical investigation on heat transfer enhancement of heat sink using perforated pin fins with inline and staggered arrangement, Journal of Applied Thermal Engineering, 125, 2017, 596-616.

[15] Maiti A., Prasad S.S., Alternative Heat Sink to Enhance Thermo-Hydraulic Behavior of an Array of Short Pin Fins, 3rd World Congress on Mechanical, Chemical, and Material Engineering, Rome, Italy, 2017.

[16] Tang, Z. G.; Liu, Q. Q.; Li, H.; Min, X. T. Numerical simulation of heat transfer characteristics of jet impingement with a novel single cone heat sink. Appl. Therm. Eng. 2017, 8, 840–852.

[17] Khattak, Z.; Ali, H.M. Air cooled heat sink geometries subjected to forced flow: A critical review. Int. J. Heat Mass Transf. 2019, 130, 141–161.

[18] Naphon, P.; Wiriyasart, S.; Arisariyawong, T.; Nakharintr, L. ANN, numerical and experimental analysis on the jet impingement nanofluids flow and heat transfer characteristics in the micro-channel heat sink. Int. J. Heat Mass Transf. 2019, 131, 329–340.

[19] Hussain, A. A.; Freegah, B.; Khalaf, B. S.; Towsyfyan, H. Numerical investigation of heat transfer enhancement in plate-fin heat sinks: Effect of flow direction and fillet profile. Case Stud. Therm. Eng. 2019, 13, 100388.