CFD Letters

Journal homepage: www.akademiabaru.com/cfdl.html
ISSN: 2180-1363

Numerical Optimization of a CPU Heat Sink Geometry

Md. Zavid Iqbal Bangalee[1,2,*], Md. Mizanur Rahman[1,2], Khairy Zaimi[3,4], Mohammad Ferdows[1,2]

1 Department of Applied Mathematics, University of Dhaka, Dhaka-1000, Bangladesh
2 Research group of fluid flow modeling and simulation, Department Applied Mathematics, University of Dhaka, Bangladesh
3 Centre of Excellence for Social Innovation & Sustainability (CoESIS), Universiti Malaysia Perlis, Perlis, Malaysia
4 Boundary Layer Research Group, Institute of Engineering Mathematics, Faculty of Applied and Human Sciences, Universiti Malaysia Perlis, Perlis, Malaysia

ARTICLE INFO

Article history:
Received 30 October 2020
Received in revised form 3 January 2021
Accepted 6 January 2021
Available online 8 March 2021

Keywords:
Heat sink; Fin; Baseplate; Turbulence Model

ABSTRACT

During its operation CPU dissipates undesirable heat. Therefore, heat sink is very essential in modern computing system to absorb extra heat dissipated by the CPU. Forced convection air cooling is common approach. In the present work, a steady-state convective heat transfer process is analyzed for CPU cooling. A rectangular fin heat sink equipped in a computer chassis is numerically investigated and simulated using the software ANSYS CFX. A two-equation based $k-\varepsilon$ turbulence model is chosen to capture the turbulence of the flow inside the domain. The overall dimension of the heat sink is optimized for three different parameters of the heat sink such as fin quantity, fin height and baseplate thickness. An optimum fin quantity, fin height and baseplate thickness are found $36, 25mm$ and $2mm$, respectively. Two different orientations of fins are also compared. Better thermal performance is achieved when the fin channel is perpendicular to the surface parallel to the outlet. The average temperature of the heat sink is found 51.5^0C. It is also predicted that the heat sink studied here is capable to keep the CPU temperature under 60^0C, that is reasonably acceptable.

1. Introduction

An Integrated Circuit (IC) used for controlling the whole computer system is known as Central Processing Unit (CPU) or simply Processor. It is considered as the brain of a microcomputer and responsible for processing most of the data and eventually generates undesirable heat. Therefore, heating problem is the major problem in modern computing systems. Overheating can easily damage the processor or hamper working efficiency. For this reason, heat removal system is necessary for cooling any CPU. At the very beginning processor was installed without heat removal system [1]. However, processor's clock speed is increasing and size is decreasing over time. So, it becomes overheated while working. For example, Intel Core i7 7700k processor dissipates 91W from the base area of 14.0625cm^2 [2] and Intel core i9 7900X deca core high performance x86 processor dissipates 140W within the area of 23.625 cm^2 [3] when operating at base frequency with all cores fully active

* *Corresponding author.*
E-mail address: zavid@du.ac.bd (Md. Zavid Iqbal Bangalee)

https://doi.org/10.37934/cfdl.13.3.115

under an Intel-defined, high-complexity workload. In these cases, their junction temperatures are 100^0C [2] and 95^0C [4] respectively.

Considering overheating challenges, many scientists have studied heat sink at different times. Free and forced convection are common approaches for cooling techniques. Besides liquid cooling techniques considering nanofluid particles, properties of nanofluid particles, heat sink geometry, fin and base thickness of the heat sink, cooling fan and form factor of CPU chassis are also investigated by many researches. Azwadi *et al.,* [5] simulated the thermal management of Advanced Technology Extended (ALX) and Balanced Technology Extended (BTX) by using ANSYS and claimed that BTX form factor is better than ATX form factor. Yu and Webb [6] used CFD (Icepak) to identify a cooling solution for desktop computer, by using an 80W CPU. It was proposed that a ducted $80mm \times 60mm$ heat sink is able to meet the CPU temperature specification. Jeremy [7] introduced a new computer cooling fan system based on computer aided modeling, computational simulation, and prototype testing. Stafford *et al.,* [8] investigated the heat transfer for flat plate using axial fan flows.

Li and Shi [9] worked on the base thickness of heat sink numerically for different convective heat transfer boundary conditions in a dimensional heat transfer model. The area ratio of a heat sink to a heat source was studied, the lowest thermal resistance was obtained and the relations among heat transfer mechanisms were discussed in their work. Tarvydas *et al.,* [10] discussed the advantages of finite element modeling and analysis of heat sinks. Huang and Chen [11] proposed a heat sink geometry to minimize the system thermal resistance of the fin array. They determined the optimal fin width and height in their work using Levenberg-Marquardt Method and the CFD-ACE+ commercial package. Researchers [12] also worked on the orientation effects of cylindrical heat sink. Kuppusamy *et al.,* [13] compared between micro channel heat sink with micro channel with triangular shape micro mixer heat sink and proposed the triangular shaped micro mixers in micro channel heat sink.

Ahmed [14] compared the thermal design between plate fin heat sink (PFHS) and ribbed plate fin heat sink (RPFHS) and concluded that the thermal performance of RPFHS is 1.55 times greater than PFHS. Romana *et al.,* [15] studied experimentally the thermal analysis of heat sink for 60W heat source. Ermangan and Rafee [16] compared the optimal geometry of a rectangular micro channel heat sink for conventional hydrophobic and super hydrophobic wall surface for low power cooling application and suggested micro channel with super hydrophobic wall surface. It was noted that micro mixer was 1.56 times better than micro channel. Mousavi *et al.,* [17] recently studied ten various configurations of interrupted, staggered and capped finned heat sink to find optimum configuration. It was noted that enough space between the fins to improve the cooling performance and capped fin enhanced the heat transfer rate. Alam *et al.,* [18] numerically studied heat transfer for a designed micro-pin-fin-heat sink using air cooling method. They found that the Nusselt number (Nu) increases with the increase in air flow velocity which enhance the heat extraction. In another study, Alam *et al.,* [19] numerically studied the effect of C-C heat flux on carbon nanotubes filled with Darcy porous medium with convective conditions. They executed computation estimation for friction factor and local Nusselt number.

To improve the thermal performance of heat sink, nanofluids are widely used. Naphon *et al.,* [20] numerically investigated the heat transfer in mini-rectangular fin heat sink for CPU of PC using de-ionized water as working fluid and claimed that will allow the design of the cooling system with improved cooling performance of the electronic equipment and increasing reliable operation of these devices. Hassan [21] studied the heat transfer inside horizontal and vertical enclosure filled with Cu-water nanofluid where a heat sink with rectangular fins represents the base plate of the enclosure. Xu *et al.,* [22] carried out numerical simulation and experiment to study the heat transfer performance of a symmetrical fractal silicon micro channel subjected to a pulsation flow. The distribution of pressure drop, temperature and the Nusselt number were presented. The influence of pulsation frequency and Reynolds number on heat transfer enhancement were also reported.

Al-Rashed *et al.,* [23] investigated the influence of nanofluids on the performance of heat sink for CPU cooling experimentally and numerically. Bahiraei and Heshmatian [24] investigated the efficiency and entropy generation of a hybrid nanofluid containing grapheme nanoplatelets decorated with silver nanoparticles in three liquid blocks for CPU cooling and suggest a liquid block for better thermal performance. Qi *et al.,* [25] experimentally studied the heat transfer and flow characteristics of nanofluids in the CPU heat sink. Comprehensive evaluation for thermo-hydraulic performance of nanofluids was investigated. It was noted that the nanpfluids with the highest nanoparticles mass fraction had not the best thermal performance and critical nanoparticle mass fraction for two different nanofluids were achieved. Han *et al.,* [26] developed a novel flat heat pipe sink with multi sink with multi-heat sources. The main operating parameter such as thermal resistance, maximum heat transfer capacity and heat dissipation capacity of the heat sink were qualitatively studied and analyzed. It was noted that overall thermal resistance of the heat sink was reduced with increasing the multi-heat source.

A lot of researches are also available on natural, forced and mixed convection. Tari and Mehrtash [27, 28] investigated natural convection from horizontal and slightly inclined plate-fin arrays using fin height, fin spacing and heat sink length and steady state natural convection from heat sink with parallel arrangement of rectangular cross section vertical plate fins on a vertical base. They suggested a set of dimensionless correlations for the convective heat transfer rate. And it is noted that at small inclinations heat transfer is almost same. Shen *et al.,* [29] studied the orientation effects on the fluid flow and heat transfer of rectangular fin heat sink under natural convection and found that denser fin arrays more sensitive to orientation. Feng *et al.,* [30] proposed a cross fin heat sink instead of conventional plate fin heat sink for overcoming thermal fluid-flow deflection. Meng *et al.,* [31] investigated the mounting angle of heat sink under natural convection and found that the heat sink was achieved the highest cooling power when its mounting angle is 90^0, and lowest for 15^0. In loop heat pipe with liquid forced convection is applied in many researches at different times. Moon *et al.,* [32] introduced a miniature heat pipe for cooling notebook PC. It was seen that junction temperature of the processor satisfies demand condition of being between 0^0C and 100^0C. Pastukhov and Maydanik [33] carried out the results of development and tests of several loop heat pipe with heat sink which are capable of sustaining and operating temperature of $72-78^0C$ on the heat source interface which dissipates $100W$ of heat. Force convection air cooling technique is very old but still very popular and widely used in personal computer. Choi *et al.,* [34] designed a new CPU cooler based on force convection to minimize the noise of fan, vibration and maximize the thermal performance. Staats and Brisson [35] investigated an air-cooled heat sink using integrated, interdigited impellers to enhance the convective heat transfer.

Dang *et al.,* [36] numerically investigated the heat transfer performance for a rack cooling system with pulsating heat pipe. It is noted that the pulsating heat pipe can be activated in less time and transfer more heat with increasing heating power. Zeng *et al.,* [37] designed an air heat sink with force convection by topology optimization and compared it with conventional heat sink and found that the topology optimized heat sink can achieve lower junction temperatures with the same pumping power. Yoon *et al.,* [38] analyzed the thermal performance of heat sink according to the partially heated surface and discussed the optimum partial heating position. They used numerical model for simulating force convection to analyze the heat transfer between heat sink and ambient air. Ozturk and Tari [39] compared three different commercial heat sinks. Also, they investigated the flow field and temperature distribution inside the chassis for force convection.

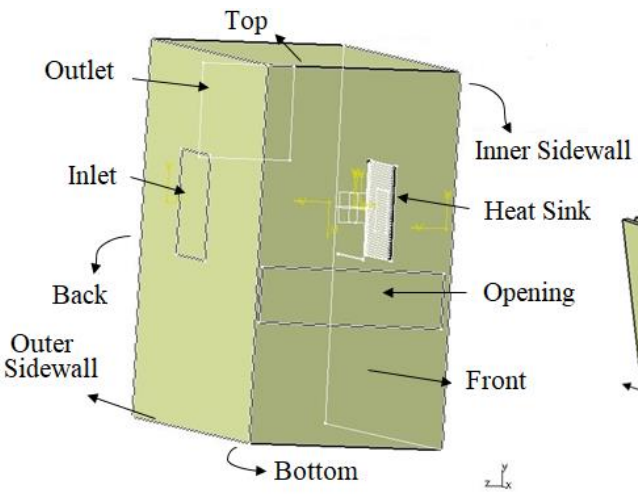
Among all types of cooling techniques described previously, forced convection air cooling including a cooling fan together with a heat sink is a common technique. In this cooling technique air is forced through the heat sink by fan and thus heat is transferred to the final medium air [40]. It is also cheap and low risk when compared to liquid cooling technique. Leakage in liquid cooling technique can damage various components of the Printed Circuit Board (PCB). Since liquid cooling needs liquid block, piping elements, tubes and cables that add more complexity. A simpler rectangular fin heat sink is considered in this study. The objective of this study is to analyze the feasibility of using a simpler heat sink to remove the CPU heat effectively and to determine the optimum geometry of the heat sink. Properties of one of the modern processor core i7 7700k is chosen. To reduce the temperature of this processor a simpler rectangular fin heat sink is numerically investigated to keep the temperature of the processor under the junction temperature in other word within the safe operating temperature.

2. Methodology

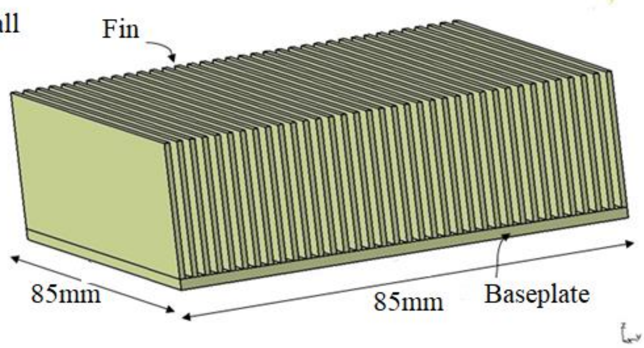
The following steps are followed to simulate the problem numerically.

2.1 Description of the Model

Computational domain in the present work is a heat sink equipped in an average size ($39.5cm \times 18cm \times 34cm$) of CPU chassis. Also, it has six sidewalls namely top, bottom, inner, outer, front and back. Inner sidewall is parallel to the xy plane. It has inlet on the outer sidewall, outlet on the back sidewall far from $20mm$ from top and $5mm$ from inner sidewall and there is an opening on the front sidewall. This opening area is about $85cm^2$. Inlet is located at the opposite sidewall of the heat sink as it can blow air directly toward the heat sink, outlet blows air outside of the chassis and opening balances the pressure gradient of air. The dimension of both inlet and outlet is about $8.5cm \times 8.5cm$. A boundary condition (heat source) is given on the bottom of the heat sink which is located at the inner sidewall. Base area of heat sink is about $85mm \times 85mm$. It has vertical and horizontal fin orientations. The model in the present work is drawn by CAD software CATIA [41] shown in Figure 1:



(a)



(b)

Fig. 1. Geometry of the computational domain. CPU chassis (a) and Heat Sink (b)

2.2 Computational Fluid Dynamics

Using computational fluid dynamics, Prajapati [42] predicted fin height of a heat sink by considering seven different cases from $0.4mm - 1.0mm$. It is found that $0.8mm$ performed better than fin heights of $0.9mm$ or $1.0mm$. Saravanakumar and Kumar [43] analyzed two different heat sink experimentally and computationally which provided physical understanding of heat transfer characteristics. Hadad *et al.,* [44] also worked on a heat sink parameter such as fin height, fin thickness and fin channel with CFD technique. Pairan *et al.,* [45] used ANSYS CFX to simulate film cooling in the leading-edge region of a turbine blade. Khan *et al.,* [46] numerically studied a steady MHD micropolar fluid flow on double stratification over a permeable a stretching/shrinking vertical sheet with the existence of chemical reaction and heat source effect. Computational fluid dynamics is very powerful tools for solving heat transfer and fluid dynamics equation. It is very popular and widely used because of its time-consuming advantages, low cost and availability of huge results. For all these reasons, the CFD technique is chosen in this study.

2.3 Governing Equations and Boundary Conditions

The governing equations of mass, momentum and energy conservation can be written in the following form.

Continuity: $\frac{\partial}{\partial x_j}(U_j) = 0; j = 1,2,3$ (1)

Momentum: $\rho U_j \frac{\partial}{x_j}(U_j) = \frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j}\left[(\mu + \mu_t)\frac{\partial U_i}{\partial x_j}\right] + \rho_0 g_i \beta(T - T_0); i = 1,2,3 \,\&\, j = 1,2,3$ (2)

Energy: $U_j \frac{\partial}{\partial x_j}(T) = \frac{\partial}{\partial x_j}\left[\left(\frac{\upsilon}{P_r} + \frac{\upsilon_t}{\sigma_t}\right)\frac{\partial T}{\partial x_j}\right]; j = 1,2,3$ (3)

Equation of state: $p = \rho RT$ (4)

The Bousinessq approximation $\rho_0 g_i \beta(T - T_0)$ is used in the momentum equation to model the buoyancy force. The transport equation for $k - \varepsilon$ turbulence model is given below,

The k equation, $\rho \frac{\partial}{\partial x_j}(U_j k) = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right] + P_k - \rho\varepsilon$ (5)

The ε equation, $\rho \frac{\partial}{\partial x_j}(U_j \varepsilon) = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_\varepsilon}\right)\frac{\partial \varepsilon}{\partial x_j}\right] + \frac{\varepsilon}{k}(C_{\varepsilon 1} P_k - C_{\varepsilon 2}\rho\varepsilon)$ (6)

Where the turbulence viscosity μ_t is linked to the turbulence kinetic energy (k) and dissipation (ε) via the relation

$\mu_t = C_\mu \rho \frac{k^2}{\varepsilon};$ (7)

Where C_μ is constant.

Assuming steady state three-dimensional flow inside the chassis all of the governing equations need to be solve and in the solid domain only energy equation needs to be solved. The property of air at 25^0C and $1atm$ are considered inside the chassis. All of the flow regimes are subsonic and airflow is assumed buoyant. Heat transfer is initiated due to thermal energy. The other boundary conditions are:

i. Inlet: $u = 0ms^{-1}$, $v = 0ms^{-1}$, $w = 5ms^{-1}$,
ii. Outlet: $u = 8ms^{-1}$, $v = 0$, $w = 0$,
iii. Opening: Opening pressure and direction with relative pressure $0\ pa$ and opening temperature is set at 25^0C,
iv. Heat flux: $64711Wm^{-2}$

2.4 Numerical Solution

ANSYS CFX-Solver Manager is used to solve the governing equation at each mesh elements of the computational domain. The following high-resolution scheme is implemented in ANSYS CFX.

$$\varphi_{ip} = \varphi_{up} + \beta\nabla\varphi \cdot \Delta r' \tag{8}$$

Where φ_{ip} is the value at upwind node and r' is the vector from the upwind node to the integration point $'ip'$. For high resolution scheme $\nabla\varphi$ is the nodal gradient of the upwind node and β is locally to be as close to 1 as possible without violating boundedness principles. It is noted that $\beta = 0$ and $\beta = 1$ represent the first and second order upwind difference scheme respectively. In the present work high resolution scheme is used. The convergent solution is achieved with the residual target 5.5×10^{-4}.

2.5 Grid Distribution

For admissible results, a well converged solution is required. And there is no substitute for fine mesh topology to get a converged solution. Therefore, ICEM CFD is used for high quality mesh generation in the present work. To get optimum mesh topology ten different types of meshes are generated within the whole computational domain. Heat sink, contact area of the processor, inlet, outlet and opening has density region of maximum elements from 2.083mm to 3.75mm and the rest of the domain has a global mesh parameter of maximum elements from 5mm to 9mm. The number of mesh elements are found from 0.9 million to 5.3 million. From the simulated results it is found that the maximum temperature of the processor is vibrated within the range from 67^0C to 71^0C for different mesh topologies. Also, it is found that for 0.9 to 1.9 million mesh elements, the solution of the system doesn't converge to the desired residuals and 2.6 million mesh elements take a large number of iterations to converge. On the other hand, 3.1 to 5.3 million mesh elements are converged within the desired residual. Among the converged solutions, better performance is obtained from 4.0 and 5.3 million mesh elements. Although 5.3 million mesh elements help to get 1.45^0C less temperature than 4.0 million mesh elements. Considering all these, the mesh topology having 4.0 million mesh elements is chosen which is shown in Figure 2 and Figure 3.

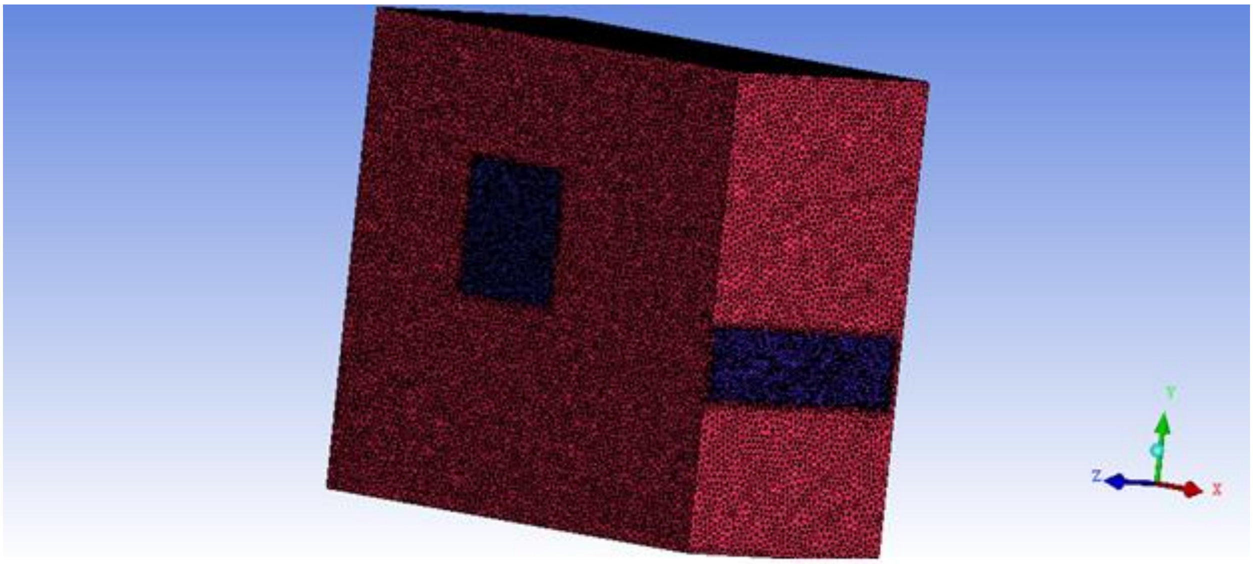
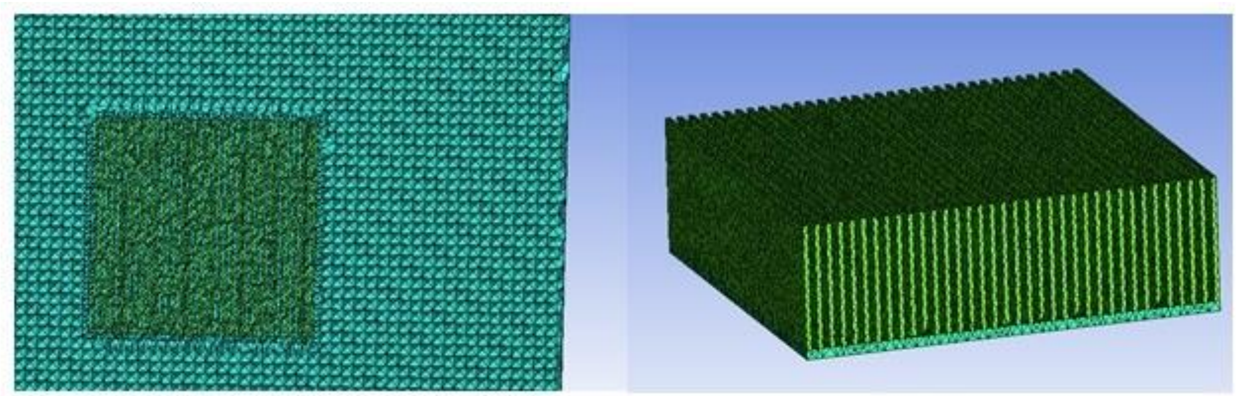


Fig. 2. Final meshed geometry in ANSYS ICEM CFD



(a) (b)

Fig. 3. (a) Density region and global mesh parameter, (b) Heat sink (Density region)

2.6 Validation of Methodology

To validate the results, a pin fin heat sink is considered which is reported by Ozturk and Tari [39]. The heat sink is drawn according to the reported geometry and the simulation is done considering the same boundary conditions. From Table 1, it is seen that the average temperature rises above ambient is reported 18.7^0 where this temperature is found 20.9^0 in our computations. The discrepancy in the temperature rise is 2.2^0 due to lack of analogy in the geometry, flow conditions and boundaries between two cases. The temperature difference between T_{max} is 3.5^0. The air velocity is not strong enough is the direct path along the fan hub and the heat sink. As a result, a hot spot is induced on the heat in the path. The larger maximum temperature in the heat sink [39] may be attributed to this fact. In this present simulation for validation, no fan is considered, that creates no hot spot. That's why our maximum temperature is reasonably lower than that of the reported results [39].

Table 1
Comparison of the temperature difference with Ozturk *et al.,* [39] for $70W$ heat source (CPU)

	Alpha PAL8952	Present Case
$T_{max}(K)$	328	324.5
$T_{min}(K)$	316	315.9
$T_{max}-T_{min}(K)$	12	8.6
Average rise above ambient, $\Delta T(K)$	18.7	20.9
Normalized non uniformity $(T_{max}-T_{max})/\Delta T$	0.64	0.41

3. Results and Discussions

A steady-state convective heat transfer process is studied numerically. The effects of various parameter of CPU heat sink are analyzed and discussed here as follows:

3.1 The Effect of Fin Quantity

Optimum number of fin is very important for a heat sink. A small quantity of fin occupies comparatively less area. Therefore, it may not transfer desired amount of heat. On the other hand, excessive number of fins have a negative impact on the convective heat transfer rate, because passage between two consecutive fins is not sufficient for fluid to flow sometimes freely. Also, for excessive number of fins, more material is used, therefore it becomes costly and heavier. For this reason, an optimum number of fins is necessary for a heat sink to remove heat efficiently. To find out the optimum number of fins, seven different 3-D models are created having 20 to 44 fins and meshes are generated accordingly by using the optimum mesh topology. In these cases, the heat sink has $2mm$ thick baseplate, $1mm$ fin width and $25mm$ fin height. The results obtained from various fin quantities and their simulated results are summarized in Table 2 and Figure 4. After increasing up to 36 fins, no significant change in temperature for further increasing fin quantity is observed and shown in Figure 4. Therefore, it can be concluded that 36 fins with $1.4mm$ fin channel is sufficient to deal with the maximum temperature of the processor effectively.

Table 2
Maximum CPU surface temperature for different number of fins

Fin quantity	Fin passage width [mm]	Temperature [^{0}C]
20	3.42	79.16
24	2.65	75.00
28	2.11	67.35
32	1.71	64.54
36	1.40	60.64
40	1.15	59.38
44	0.95	59.37

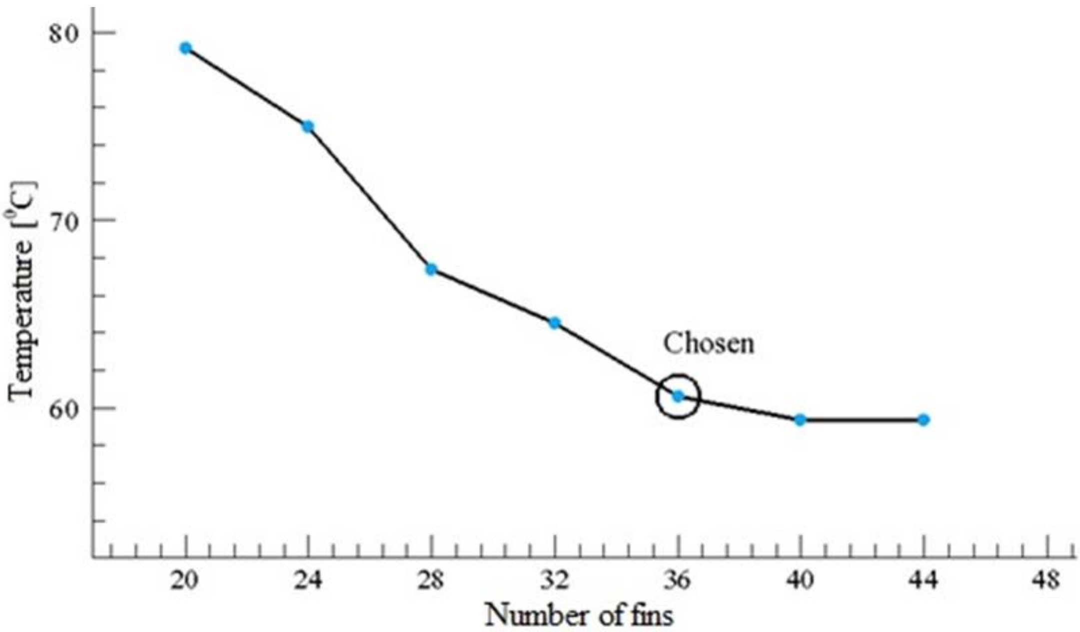


Fig. 4. Maximum CPU surface temperature for different fin quantities

3.2 The Effect of Baseplate Thickness

Baseplate thickness is another important parameter for a heat sink. Six different 3D models are drawn and meshes are generated for $1mm$ to $3.5mm$ baseplate thickness. In all cases, heat sink has 36 fins and $25mm$ fin height. Optimum mesh topology is used for simulation and simulated results are gathered for various baseplate thickness. Results showed that from 1mm to $3.5mm$ baseplate thickness, maximum temperature of the processor is decreasing. But more thickness is responsible for more weight and need more material cost to build. Therefore, 2mm baseplate thickness is more suitable and it is selected as optimum for further improvisation of the heat sink geometry. The results obtained from various fin quantities and their simulated results are summarized in Table 3 and Figure 5.

Table 3
Maximum CPU surface temperature for different baseplate thickness

Baseplate thickness (mm)	Temperature (^{0}C)	Baseplate thickness (mm)	Temperature (^{0}C)
1	63.90	2.5	59.99
1.5	61.79	3	58.88
2	60.64	3.5	58.21

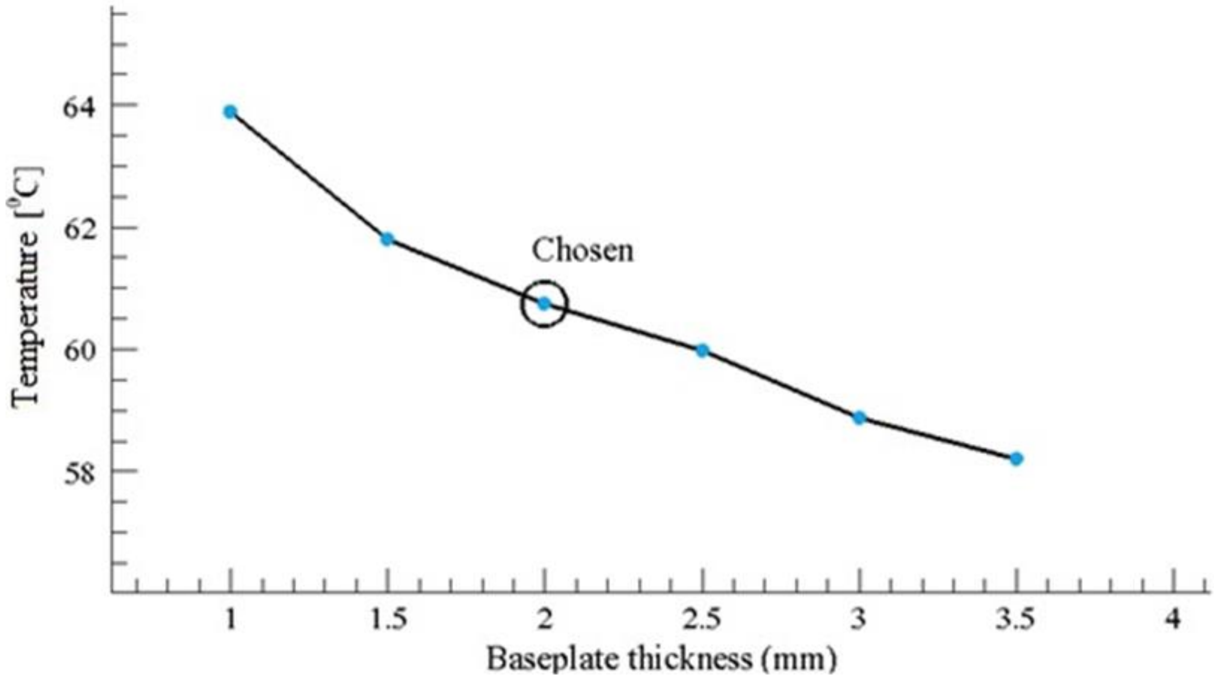


Fig. 5. Maximum CPU surface temperature for various baseplate thicknesses

3.3 The Effect of Fin Height

The next attention is to optimize the height of the fins. CPU chassis has limitation in spaces. Therefore, fin height cannot be enlarged as long as possible. Optimum fin height is required to deal with the space limitation and maximum temperature of the processor. To find out the optimum height several 3D geometries are created with 10mm to 50mm fin height. In these cases, the quantity of fin [36], the thickness of baseplate were unchanged and optimum mesh topology is used. Simulated results are shown in Table 4. It is found that maximum CPU surface temperature decreases as the fin height increases as shown in Figure 6. However, the temperature change is found less significant for fin height larger than $25mm$. It may happen due to the surface area adjacent to the given boundary condition.

Table 4
Maximum CPU surface temperature for different fin height

Fin height (mm)	Temperature (^{0}C)
10	95.63
15	77.03
20	67.62
25	60.64
30	55.65
40	52.74
50	49.23

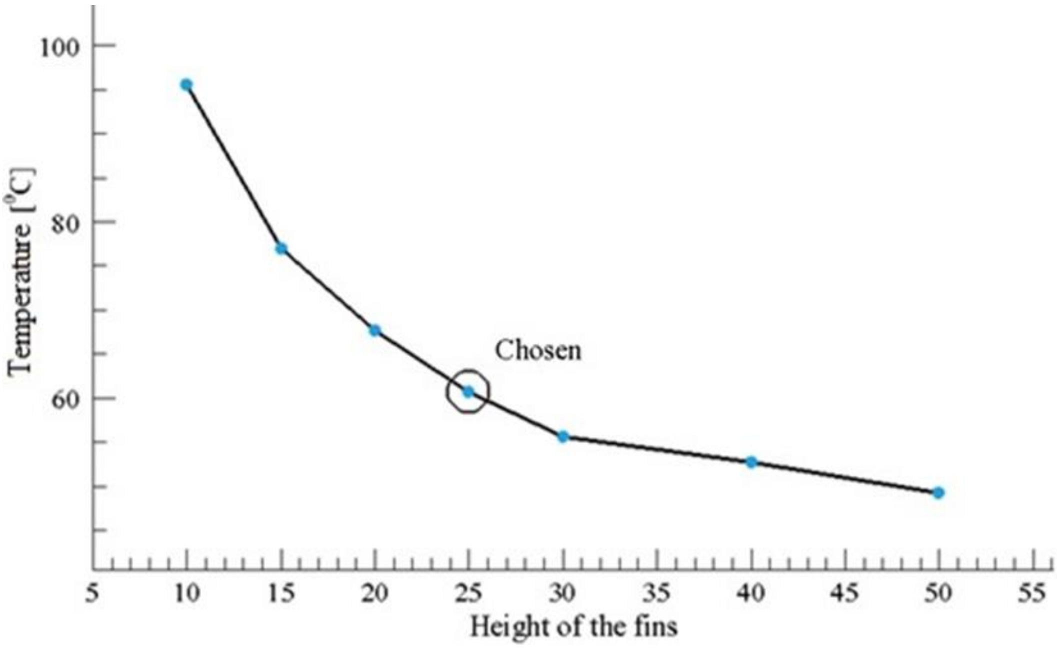


Fig. 6. Maximum CPU surface temperature for various fin height

3.4 The Effect of Fin Orientation

Present work contains a rectangular fin heat sink for forced convection. As mentioned earlier, in forced convection air is flowed through fin channel. Therefore, orientation of fin also has an impact on overall thermal performance of the heat sink. Two 3-D models considering horizontal and vertical orientations of fin are simulated. It is found that horizontal fin orientation shows better thermal performance than that of vertical fin orientation. It is attributed to the outlet position. In the case of vertical fin orientation, outlet is located at perpendicular to the fin channel and air cannot flow directly toward the outlet after passing through the fin channel. On the other hand, in the case of horizontal fin, air flow moves directly toward the surface parallel to the outlet after passing through the fin channel. Therefore, better thermal performance is achieved. For both cases simulated results are shown in Table 5.

Table 5
Maximum CPU surface temperature for fin orientation

Orientation of fin	Temperature (^{0}C)
Horizontal	60.64 (Chosen)
Vertical	64.98

3.5 Temperature Distribution

Numerical computations are carried out for a $91W$ CPU (maximum load). Considering all of the analysis, a general suitable case is set for this study. In this case, the fin quantiy, fin height and baseplate thickness are found 36, $25mm$ and $2mm$, respectively. The temperature distribution for heat sink obtained from the computation is shown in Figure 7 and Figure 8. It is found that the temperature is higher in the centre of the heat sink than that of close to the sides. Heat is properly distributed along the fins. The maximum temperature of the CPU surface is calculated 59.64^0C and the average temperature of the heat sink is calculated 51.5^0C.

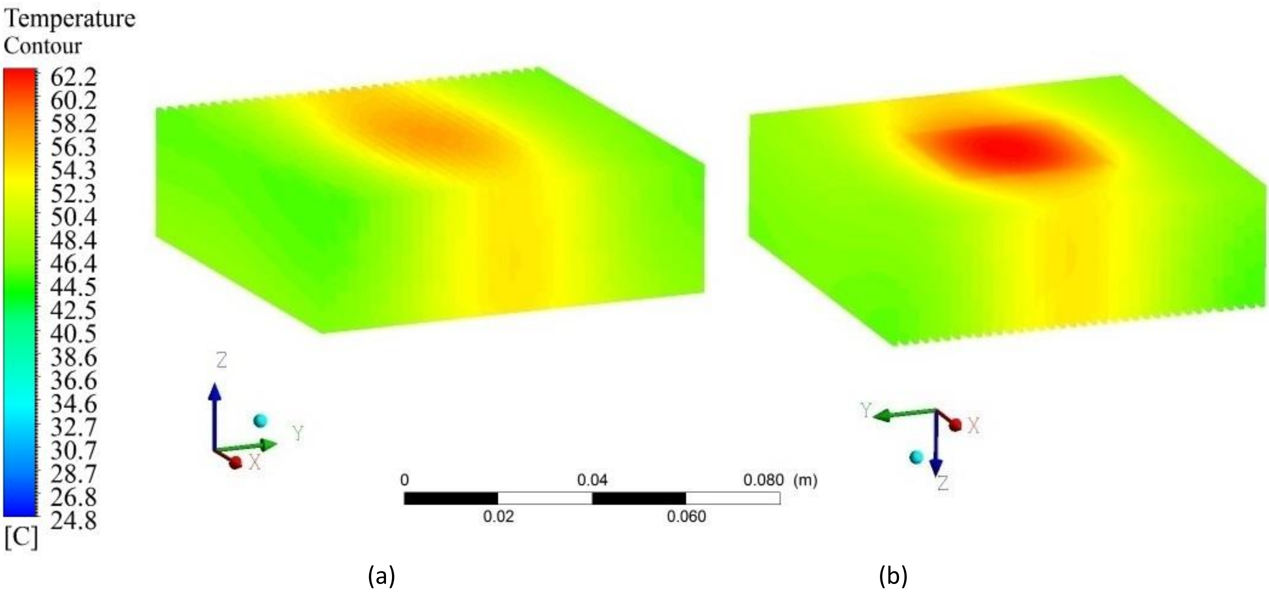


Fig. 7. Temperature distribution of the heat sink. Top view (a) and bottom view (b)

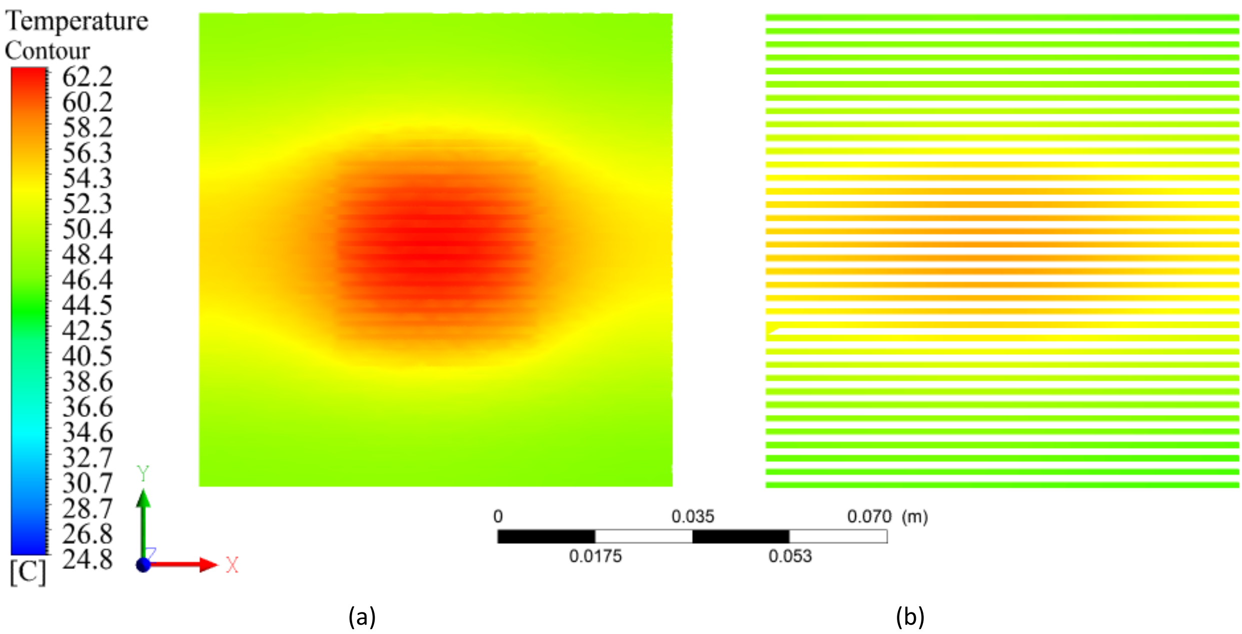


Fig. 8. Temperature distribution of baseplate (a) and fin tip (b) of the heat sink

4. Conclusions

A rectangular fin heat sink is drawn with CATIA and then the domain is discretized using ICEM CFD. The governing equation at each grid is solved by the software ANSYS CFX, to determine whether it is possible to deal with the junction temperature of the high-performance processor. Different parameters of the heat sink are optimized for better thermal performance. The temperature and flow distribution are also investigated numerically. A two-equation based turbulence model is applied to capture the turbulence of the flow field. The following conclusions can be drawn from the study:

i. Although fin is responsible for increasing surface area that also increases the cooling rate of the heat sink, but an excessive number of fin blocks the flow path that disrupts the convection rate.
ii. When the baseplate thickness is increased, the overall area of the heat sink is also increased. Therefore, the conduction is increased and thus the heat sink is capable to absorb more heat.
iii. When fin height is increased, surface area is also increased, however after reaching the certain height there is no significant change in the maximum temperature. Therefore, a moderate height of fin is desired.
iv. Better thermal performance is achieved if the fin channel is perpendicular to the surface parallel to the outlet.

Present work has been completed within a time limitation. As unsteady simulation needs more computational time this paper only examines steady-state computations. However, for more accurate results unsteady computations are necessary. A further study on this topic may be done considering the unsteady computation.

Acknowledgement
The author would like to acknowledge the support from the Fundamental Research Grant Scheme (FRGS) under a grant number of FRGS/1/2018/STG06/UNIMAP/02/3 from the Ministry of Education, Malaysia. This work was carried out using the computational facilities available in the Department of Applied Mathematics, University of Dhaka, Bangladesh. The University Grants Commission (UGC) in Bangladesh and Bose Centre for Advanced Study and Research in Natural Sciences, University of Dhaka, Bangladesh funded it partially.

References
[1] Magadum, Prasannakeerti Prakash, and Gaikwad Mahesh Shamrao. "Analysis & Optimization of Processor Cooling Fins.". https://doi.org/10.17605/OSF.IO/E5WGJ
[2] Processor core i7 7700K.
[3] Processor core i9 7900X.
[4] Processor core i9 7900X.
[5] Nor Azwadi, C. S., C. K. Goh, and M. Y. Afiq Witri. "Computational analysis of heat flow in computer casing." In *AIP Conference Proceedings*, vol. 1440, no. 1, pp. 199-208. American Institute of Physics, 2012. https://doi.org/ 10.1063/1.4704218
[6] Yu, C-W., and R. L. Webb. "Thermal design of a desktop computer system using CFD analysis." In *Seventeenth Annual IEEE Semiconductor Thermal Measurement and Management Symposium (Cat. No. 01CH37189)*, pp. 18-26. IEEE, 2001. https://doi.org/ 10.1109/STHERM.2001.915139
[7] Li, Jeremy Zheng. "Computer Aided Modeling and Simulation of Cooling Fan System."
[8] Stafford, Jason, Ed Walsh, Vanessa Egan, and Ronan Grimes. "Flat plate heat transfer with impinging axial fan flows." *International Journal of Heat and Mass Transfer* 53, no. 25-26 (2010): 5629-5638. https://doi.org/10.1016/j.ijheatmasstransfer.2010.08.020
[9] Li, Ji, and Zhong-shan Shi. "3D numerical optimization of a heat sink base for electronics cooling." *International communications in heat and mass transfer* 39, no. 2 (2012): 204-208. https://doi.org/10.1016/j.icheatmasstransfer.2011.12.001
[10] Tarvydas, Paulius, Alius Noreika, and Z. Staliulionis. "Analysis of heat sink modelling performance." *Elektronika Ir Elektrotechnika* 19, no. 3 (2013): 43-46. https://doi.org/10.5755/j01.eee.19.3.3695
[11] Huang, Cheng-Hung, and Yu-Hsiang Chen. "An impingement heat sink module design problem in determining simultaneously the optimal non-uniform fin widths and heights." *International Journal of Heat and Mass Transfer* 73 (2014): 627-633. https://doi.org/10.1016/j.ijheatmasstransfer.2014.02.026

[12] Jang, Daeseok, Seung-Jae Park, Se-Jin Yook, and Kwan-Soo Lee. "The orientation effect for cylindrical heat sinks with application to LED light bulbs." *International Journal of Heat and Mass Transfer* 71 (2014): 496-502. https://doi.org/10.1016/j.ijheatmasstransfer.2013.12.037
[13] Jang, Daeseok, Seung-Jae Park, Se-Jin Yook, and Kwan-Soo Lee. "The orientation effect for cylindrical heat sinks with application to LED light bulbs." *International Journal of Heat and Mass Transfer* 71 (2014): 496-502. https://doi.org/10.1016/j.ijheatmasstransfer.2015.07.088
[14] Ahmed, Hamdi E. "Optimization of thermal design of ribbed flat-plate fin heat sink." *Applied Thermal Engineering* 102 (2016): 1422-1432. https://doi.org/10.1016/j.applthermaleng.2016.03.119
[15] Romana, Syeda, S. Shahar Banu, Md Irfan Ali, MA Mujeeb Iqbal, and S. Irfan Sadaq. "Thermal Analysis of Heat Sink." *Materials Today: Proceedings* 5, no. 2 (2018): 5481-5486. https://doi.org/10.1016/j.matpr.2017.12.137
[16] Ermagan, Hamidreza, and Roohollah Rafee. "Geometric optimization of an enhanced microchannel heat sink with superhydrophobic walls." *Applied Thermal Engineering* 130 (2018): 384-394. https://doi.org/10.1016/j.applthermaleng.2017.11.039
[17] Mousavi, Hamed, A. Ali Rabienataj Darzi, Mousa Farhadi, and Mohamad Omidi. "A novel heat sink design with interrupted, staggered and capped fins." *International Journal of Thermal Sciences* 127 (2018): 312-320. https://doi.org/10.1016/j.ijthermalsci.2018.02.003
[18] Alam, Mir Waqas, Suvanjan Bhattacharyya, Basma Souayeh, Kunal Dey, Faicel Hammami, Mohammad Rahimi-Gorji, and Ranjib Biswas. "CPU heat sink cooling by triangular shape micro-pin-fin: Numerical study." *International Communications in Heat and Mass Transfer* 112 (2020): 104455. https://doi.org/10.1016/j.icheatmasstransfer.2019.104455
[19] Alam, Mir Waqas, Alaaedeen Abuzir, Basma Souayeh, Essam Yasin, Najib Hdhiri, and Fayçal Hammami. "Theoretical analysis of carbon nanotubes (SWCNT/MWCNT) over a Wang's stretching sheet under CC heat flux." *Physica Scripta* 95, no. 10 (2020): 105207. https://doi.org/10.1088/1402-4896/abb563
[20] Naphon, Paisarn, Setha Klangchart, and Somchai Wongwises. "Numerical investigation on the heat transfer and flow in the mini-fin heat sink for CPU." *International Communications in Heat and Mass Transfer* 36, no. 8 (2009): 834-840. https://doi.org/10.1016/j.icheatmasstransfer.2009.06.010
[21] Hassan, Hamdy. "Heat transfer of Cu–water nanofluid in an enclosure with a heat sink and discrete heat source." *European Journal of Mechanics-B/Fluids* 45 (2014): 72-83. https://doi.org/10.1016/j.euromechflu.2013.12.003
[22] Xu, Shanglong, Weijie Wang, Kuang Fang, and Chun-Nam Wong. "Heat transfer performance of a fractal silicon microchannel heat sink subjected to pulsation flow." *International Journal of Heat and Mass Transfer* 81 (2015): 33-40. https://doi.org/10.1016/j.ijheatmasstransfer.2014.10.002
[23] Al-Rashed, Mohsen H., Grzegorz Dzido, Mateusz Korpyś, Jacek Smołka, and Janusz Wójcik. "Investigation on the CPU nanofluid cooling." *Microelectronics Reliability* 63 (2016): 159-165. https://doi.org/10.1016/j.microrel.2016.06.016
[24] Bahiraei, Mehdi, and Saeed Heshmatian. "Efficacy of a novel liquid block working with a nanofluid containing graphene nanoplatelets decorated with silver nanoparticles compared with conventional CPU coolers." *Applied Thermal Engineering* 127 (2017): 1233-1245. https://doi.org/10.1016/j.applthermaleng.2017.08.136
[25] Qi, Cong, Jinding Hu, Maoni Liu, Leixin Guo, and Zhonghao Rao. "Experimental study on thermo-hydraulic performances of CPU cooled by nanofluids." *Energy Conversion and Management* 153 (2017): 557-565. https://doi.org/10.1016/j.enconman.2017.10.041
[26] Han, Xiaoxing, Yaxiong Wang, and Qianqing Liang. "Investigation of the thermal performance of a novel flat heat pipe sink with multiple heat sources." *International Communications in Heat and Mass Transfer* 94 (2018): 71-76. https://doi.org/10.1016/j.icheatmasstransfer.2018.03.017
[27] Tari, Ilker, and Mehdi Mehrtash. "Natural convection heat transfer from horizontal and slightly inclined plate-fin heat sinks." *Applied Thermal Engineering* 61, no. 2 (2013): 728-736. https://doi.org/10.1016/j.applthermaleng.2013.09.003
[28] Tari, Ilker, and Mehdi Mehrtash. "Natural convection heat transfer from inclined plate-fin heat sinks." *International journal of heat and mass transfer* 56, no. 1-2 (2013): 574-593. https://doi.org/10.1016/j.ijheatmasstransfer.2012.08.050
[29] Shen, Qie, Daming Sun, Ya Xu, Tao Jin, and Xu Zhao. "Orientation effects on natural convection heat dissipation of rectangular fin heat sinks mounted on LEDs." *International Journal of heat and mass transfer* 75 (2014): 462-469. https://doi.org/10.1016/j.ijheatmasstransfer.2014.03.085
[30] Feng, Shangsheng, Meng Shi, Hongbin Yan, Shanyouming Sun, Feichen Li, and Tian Jian Lu. "Natural convection in a cross-fin heat sink." *Applied Thermal Engineering* 132 (2018): 30-37. https://doi.org/10.1016/j.applthermaleng.2017.12.049

[31] Meng, Xiangrui, Jie Zhu, Xinli Wei, and Yuying Yan. "Natural convection heat transfer of a straight-fin heat sink." *International Journal of Heat and Mass Transfer* 123 (2018): 561-568. https://doi.org/10.1016/j.ijheatmasstransfer.2018.03.002

[32] Moon, Seok Hwan, Gunn Hwang, Ho Gyeong Yun, Tae Goo Choy, and Young Il Kang. "Improving thermal performance of miniature heat pipe for notebook PC cooling." *Microelectronics Reliability* 42, no. 1 (2002): 135-140. https://doi.org/10.1016/S0026-2714(01)00226-8

[33] Pastukhov, Vladimir G., and Yury F. Maydanik. "Low-noise cooling system for PC on the base of loop heat pipes." In *Twenty-Second Annual IEEE Semiconductor Thermal Measurement And Management Symposium*, pp. 95-101. IEEE, 2006. https://doi.org/10.1109/STHERM.2006.1625212

[34] Choi, Jeehoon, Minjoong Jeong, Junghyun Yoo, and Minwhan Seo. "A new CPU cooler design based on an active cooling heatsink combined with heat pipes." *Applied thermal engineering* 44 (2012): 50-56. https://doi.org/10.1016/j.applthermaleng.2012.03.027

[35] Staats, Wayne L., and J. G. Brisson. "Active heat transfer enhancement in air cooled heat sinks using integrated centrifugal fans." *International Journal of Heat and Mass Transfer* 82 (2015): 189-205. https://doi.org/10.1016/j.ijheatmasstransfer.2014.10.075

[36] Dang, Chao, Li Jia, and Qianyi Lu. "Investigation on thermal design of a rack with the pulsating heat pipe for cooling CPUs." *Applied Thermal Engineering* 110 (2017): 390-398. https://doi.org/10.1016/j.applthermaleng.2016.08.187

[37] Zeng, Shi, Bugra Kanargi, and Poh Seng Lee. "Experimental and numerical investigation of a mini channel forced air heat sink designed by topology optimization." *International Journal of Heat and Mass Transfer* 121 (2018): 663-679. https://doi.org/10.1016/j.ijheatmasstransfer.2018.01.039

[38] Yoon, Youngchan, Seung-Jae Park, Dong Rip Kim, and Kwan-Soo Lee. "Thermal performance improvement based on the partial heating position of a heat sink." *International Journal of Heat and Mass Transfer* 124 (2018): 752-760. https://doi.org/10.1016/j.ijheatmasstransfer.2018.03.080

[39] Ozturk, Emre, and Ilker Tari. "Forced air cooling of CPUs with heat sinks: A numerical study." *IEEE transactions on components and packaging technologies* 31, no. 3 (2008): 650-660. https://doi.org/10.1109/TCAPT.2008.2001840

[40] Carr, Jason D. "An Examination of CPU Cooling Technologies: Thermal Management Option for Engineers." *DSI, dsiventures. com. www. liquid-cooling. org/wp.../05/CPU-Cooling-Technologies. pdf* (2018).

[41] CATIA

[42] Prajapati, Yogesh K. "Influence of fin height on heat transfer and fluid flow characteristics of rectangular microchannel heat sink." *International Journal of Heat and Mass Transfer* 137 (2019): 1041-1052. https://doi.org/10.1016/j.ijheatmasstransfer.2019.04.012

[43] Saravanakumar, T., and D. Senthil Kumar. "Performance analysis on heat transfer characteristics of heat SINK with baffles attachment." *International Journal of Thermal Sciences* 142 (2019): 14-19. https://doi.org/10.1016/j.ijthermalsci.2019.04.002

[44] Hadad, Yaser, Bharath Ramakrishnan, Reza Pejman, Srikanth Rangarajan, Paul R. Chiarot, Arvind Pattamatta, and Bahgat Sammakia. "Three-objective shape optimization and parametric study of a micro-channel heat sink with discrete non-uniform heat flux boundary conditions." *Applied Thermal Engineering* 150 (2019): 720-737. https://doi.org/10.1016/j.applthermaleng.2018.12.128

[45] Pairan, Mohamad Rasidi, N. Asmuinb, and Hamidon Salleh. "Simulation of Film Cooling in the Leading Edge Region of a Turbine Blade Using ANSYS CFX." *Journal of Advanced Research in Fluid Mechanics and Thermal Sciences* 1, no. 1 (2014): 1-10.

[46] Khan, Ansab Azam, Suliadi Firdaus Sufahani, Khairy Zaimi, and Mohammad Ferdows. "MHD Flow and Heat Transfer of Double Stratified Micropolar Fluid over a Vertical Permeable Shrinking/Stretching Sheet with Chemical Reaction and Heat Source." *Journal of Advanced Research in Applied Sciences and Engineering Technology* 21, no. 1 (2020): 1-14. https://doi.org/10.37934/araset.21.1.114