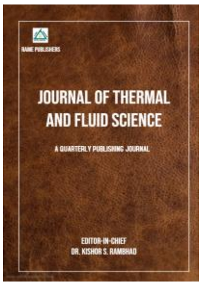
Journal of Thermal and Fluid Science
Volume 3, Issue 1, June 2022, pp 32 - 37
https://doi.org/10.26706/jtfs.3.1.arset7157



Simulation of Micro-Chip Heat Sink to Investigate the Thermo-Fluid Behaviour and Temperature Distribution

Debasish Biswas[1]
debasishmechanical@yahoo.co.in

Krishna Ladha[2]
krishna.ladha18@gmail.com

Aditya Deb[2]
aditya2016deb@gmail.com

[1]Assistant Professor, Department of Mechanical Engineering, Kalyani Government Engineering College, Kalyani-741235, India

[2]Student, Department of Mechanical Engineering, Kalyani Government Engineering College, Kalyani-741235, India

***Abstract*— The efficient cooling from heat sinks is important for the proper functioning and longevity of a central processing unit (CPU). In this Project, ANSYS simulation of a modified micro-fin heat sink was done and the results were analyzed to get an idea of the approximate rate of Heat transfer, Temperature distribution and Thermo fluid behavior of the heat sink mounted on the central processing unit. Air cooling methods by free convection and conduction are used for heat extraction. The simulation was based on the effects on inlet fluid velocity, design of fins and location of source on the performance of the Heat Sink. The Turbulent SST model is used. The Heat transfer rate, temperature and pressure distribution were obtained for different designs of fin. After complete analysis, an increased air flow velocity at inlet and thin sheet fins with source at the bottom of heat sink provides the most effective heat transfer rate.**

Keywords— Micro-Chip Heat Sink, Heat Transfer, Temperature Distribution, Ansys, Turbulent SST Model

I. Introduction

With the advent of modern technology, computers have become very important in our day-to-day life. Now a day, computers are capable of transferring high speed data leading to increased heat dissipation and higher heat density [1]. This results in hotspots in microprocessor chip. Hot spot produces locally high temperatures and huge temperature gradients through the microprocessor chip, which poorly affects the microprocessor consistency and performance [2]. Thus, it is very much necessary to cool the chip so that it can work efficiently [3]. The small size of chip makes it very much complex as well as difficult to dissipate the heat effectively [4]. A numerous methods are available for cooling electronic components used in industry, such as heat pipe and jet impingement cooling [5]. Air and other liquids can be used as medium to cool the chip. However, liquid cooling methods are complex in design, expensive and need regular maintenance [6]. So, it is necessitated to instill the effectiveness of air-cooling technologies. Therefore, the present study is to investigate the HT performance of Micro fin heat sink in cooling of the main board chip of a CPU [7].

II. Geometry

The primary objective is concerned about the effective cooling of microchip of CPU by forced convection. For the purpose a rectangular Fluid Domain of length 21mm and width 13mm is taken in consideration with the inlet and outlet opening resembling a square of side 4 mm projected 3.5 mm from the rectangular domain. Keeping the above geometry constant, the fins geometry was modified as shown:

Technical Article
Published online – 09 June 2022

Cite this article – Debasish Biswas, Krishna Ladha, Aditya Deb, "Simulation of Micro-Chip Heat Sink to Investigate the Thermo-Fluid Behaviour and Temperature Distribution", *Journal of Thermal and Fluid Science*, RAME Publishers, vol.3, issue 1, pp. 32-37, 2022.
https://doi.org/10.26706/jtfs.3.1.arset7157

A. Cuboidal Fins

Fin Details:

Base Dimensions:

Length = 2 mm Width = 2 mm Height = 3 mm

Distance between fins = 5 mm

Location of fins = 2 mm on either side about the centerline of the rectangular fluid domain

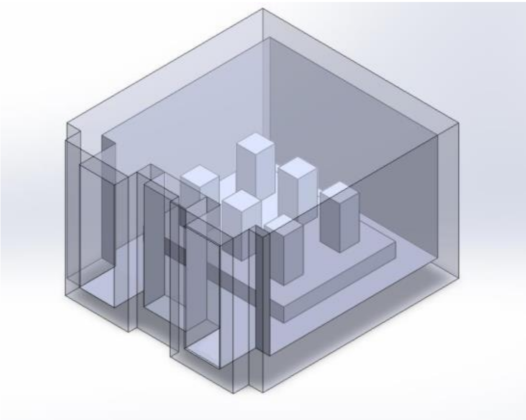


Figure 1. Cuboidal Fins

B. Cylindrical Fins

Fin Details:

Base Dimensions: Diameter = 1mm Height =1.5 mm

Distance between fins = 5 mm

Location of fins = 5 mm on both side about the centerline of the rectangular fluid domain and 4 mm on the centerline.

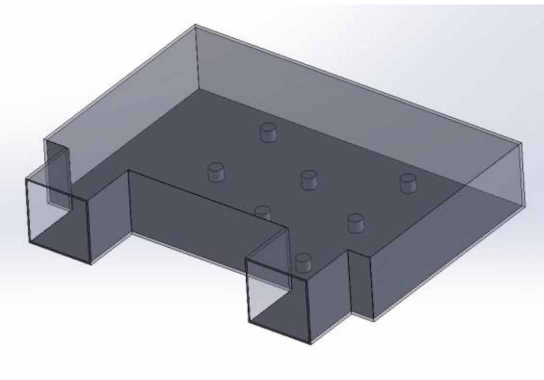


Figure 2. Cylindrical Fins

C. Thin Sheet Fins with Source in Between

Fin Details:

Base Dimensions:

Length = 1mm Width = 0.1mm Height =2 mm

Distance between fins = 0.5 mm along the length of fluid domain and 1.3 mm along the width

Location of fins = On either side about the centerline 2 mm along the length and 1 mm along width of the rectangular fluid domain.

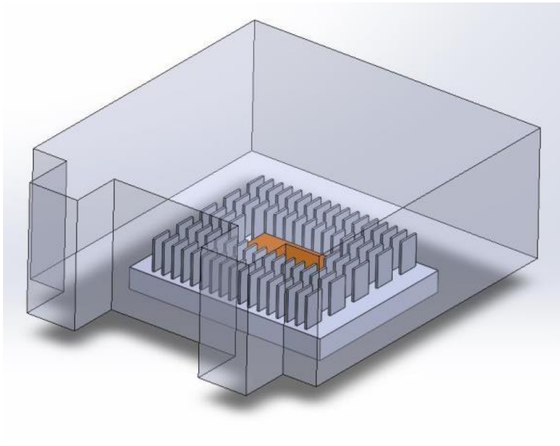


Figure 3. Thin Sheet Fins with Source in Between

D. Thin Sheet Fins with Source Below Heat Sink

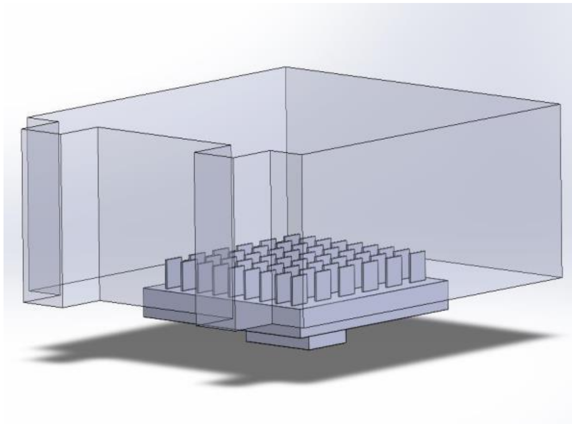


Figure 4. Thin Sheet Fins with Source Below Heat Sink

Fin Details:

Base Dimensions:

Length = 1mm Width = 0.1 mm Height = 2 mm

Distance between fins = 0.5 mm along the length of fluid domain and 1.3 mm along the width Location of fins =0.65 mm on either side about the centerline the rectangular fluid domain Considering the maximum effective cooling via forced convective cooling and faster flow of fluid inside the fluid domain, resulting in faster cooling of the microchip, the final modified design of fin i.e. “Thin Sheet Fins with Source below Heat Sink” was considered best fit for further analysis.

III. SIMULATION AND ANALYSIS

For the simulation ANSYS WORKBENCH 2021 is employed. Using Fluid Flow (Fluent) the 3D model of the selected microchip heat sink is imported and grid is generated using meshing.

A. Fluid Domain Mesh Design

Element size = 1 mm

Element behavior = smooth

Nodes = 1040

Number of elements = 708

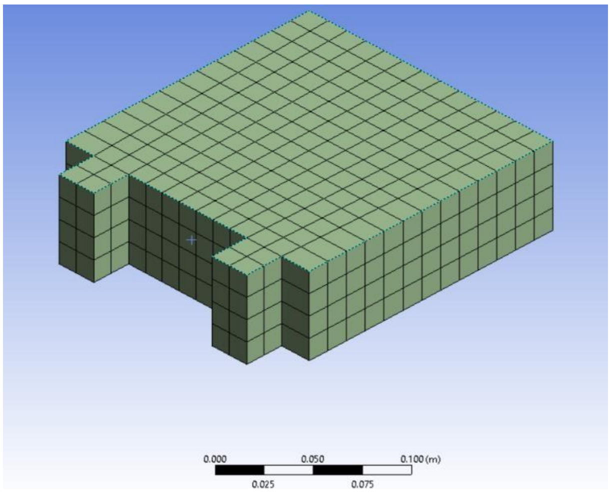


Figure 5. Mesh Design for Fluid Domain

B. Heat Sink Mesh Design

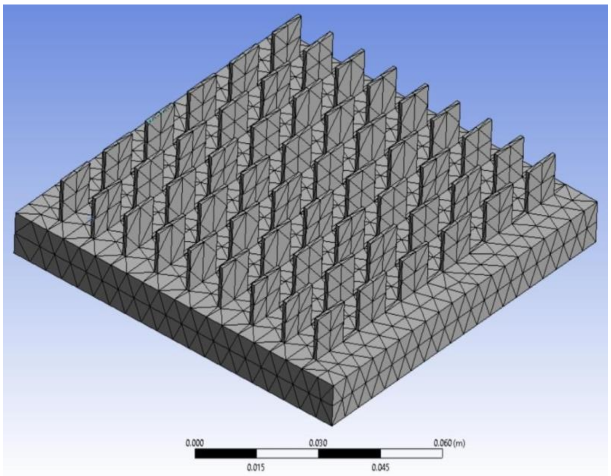


Figure 6. Mesh Design for Heat Sink

Element size = 0.5 mm

Element behavior = course

Nodes = 2369

Number of elements = 8067

C. Chip Sink Mesh Design

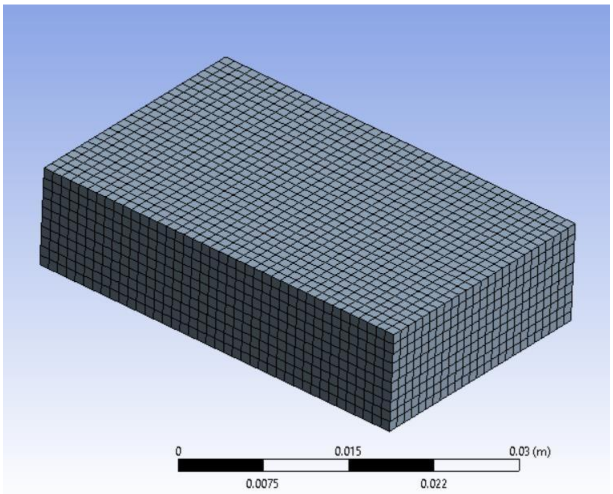


Figure 7. Mesh Design of Chip Sink

Element size = 0.1 mm

Element behavior = course

Nodes = 11726

Number of elements = 10000

D. All Domain Mesh Design

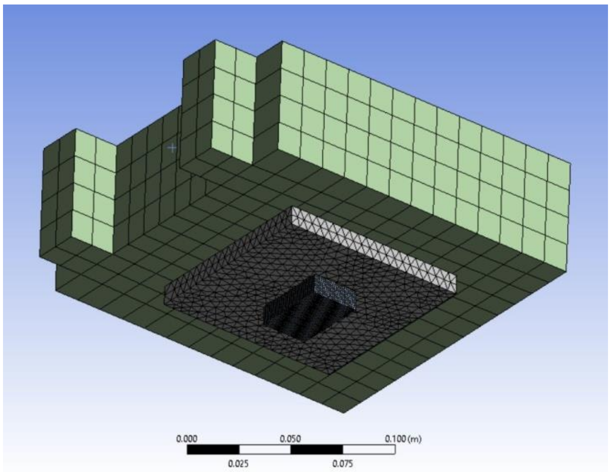


Figure 8. Mesh Design of All Domain

Nodes = 15135

Number of elements = 18775

IV. MODEL SETUP

The governing equations are discretized on a uniform structured grid using finite volume method. Energy equation is implied. Transition SST model has been used. Variables of the wall are computed as scalable wall functions. The inlet temperature of the working fluid is 300 K and a uniform wall heat flux of 2.0 kW/m^2 is imposed on the heating plate. Velocity at inlet is specified to 3 m/s. The convergence criterion for momentum, continuity and energy are set at 10^{-5}, 10^{-6} and 10^{-7}, respectively. The convergence criterion for the turbulence quantities was fixed at 10^{-4}. Initializing the setup to hybrid, calculations were computed to obtain desired results.

V. RESULTS AND DISCUSSIONS

After running simulation in Ansys Workbench 2021, the following results were received as displayed:

A. Temperature Distribution

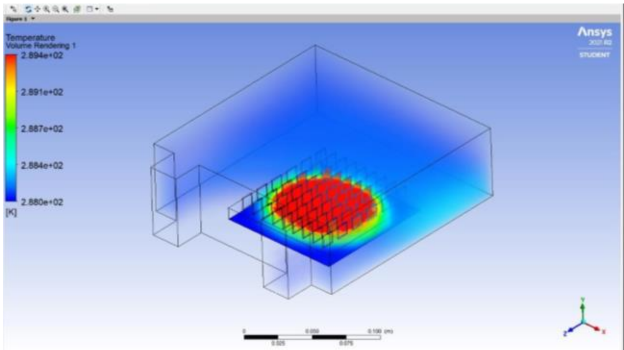


Figure 9. Temperature Distribution

A contour on the upper surface of the heat sink was generated; number of contours is assigned to 500. The variable is specified to temperature. The temperature range kept from 288 K to 290 K. The volume rendering is generated throughout the fluid domain with temperature range kept same as above, to depict the temperature distribution. Resolution is specified to 50 and transparency kept to 0.2. The temperature raises steeply up to 289.4 K near the heat sink (represented by the bright red colour) and gradually decreases towards the periphery of the sink. The fluid only heats up a little (288.4 K) above the sink and near the outlet due to the transfer of heat via convection (represented by the light blue colour). Not much difference in temperature in the fluid domain was observed as such (inky blue colour) depicting constant temperature.

B. Pressure Distribution

Volume rendering generated throughout the fluid domain. Variable is selected to Pressure with 50 resolutions and 0.2 transparencies. Range was specified to local settings. The pressure is evenly distributed inside the fluid domain depicting the proper fluid flow inside. The Pressure is highest near the outlet wall (bright red colour 8.143 Pa). Pressure is lower in between the inlet and outlet region (light blue colour 0.7898 Pa) due to minimal fluid movement.

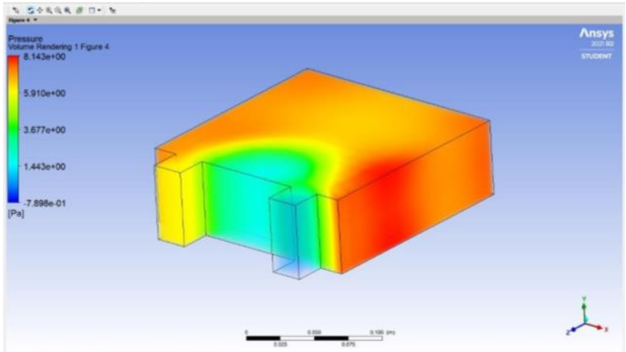


Figure 10. Pressure Distribution

C. Temperature Streamline

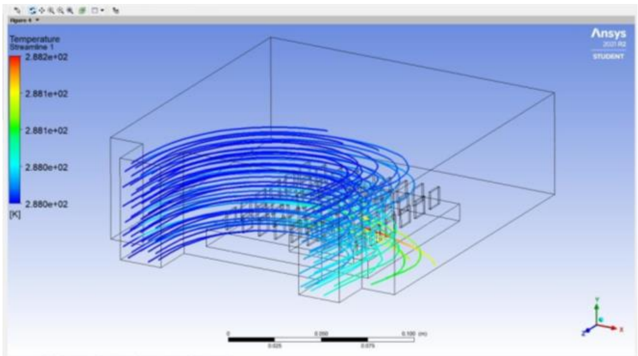


Figure 11. Temperature Streamline

3D Streamline generated from the inlet with temperature as variable, ranging from 288 K to 288.2 K was made to visualize the heat carried away by the convection currents. 50 equally spaced points were used. The inlet temperature is low (Inky blue line). Above the heat sink the fluid stream absorbs hence a rise in temperature in observed (Bright red line). Near the outlet a little heated up stream (Light blue line) leaves the region.

D. Velocity Streamline

3D streamline generated from the inlet with velocity as variable, ranging from 0.00113 m/s to 2.953 m/s were made to study the velocities of the fluid streams. 50 equally spaced points were used. In the inlet region the velocity of the fluid flow is high (Bright red line) near the centre of the opening and decreases towards the wall (Inky blue line). The flow velocities for different streams are maintained constant throughout the fluid domain.

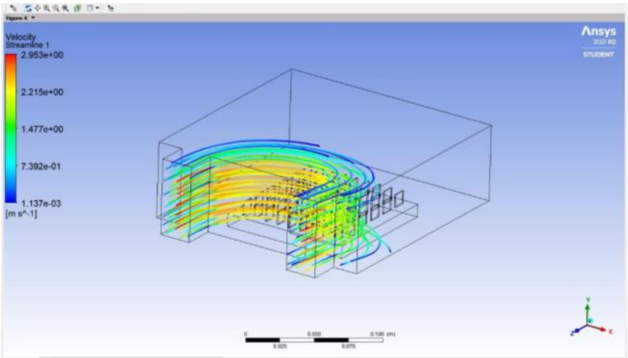


Figure 12. Velocity Streamline

E. Fluid Flow Direction (Velocity Vectors)

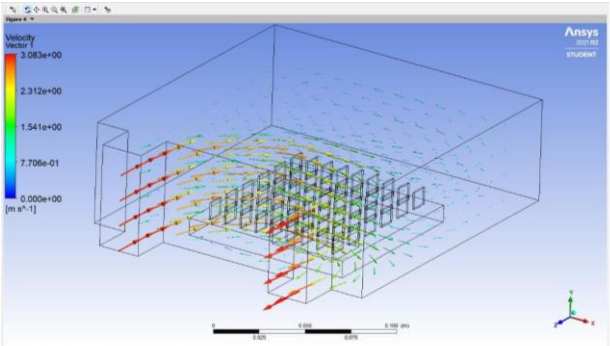


Figure 13. Velocity Vectors

Track of direction of fluid flow was achieved using Vector command using variable as Velocity. 3D arrows used as symbols. Near the inlet and outlet, the fluid reaches peak velocity (Bright red arrows) and is lowest at backside of the heat sink region (Deep blue arrows). Near the openings the flow is in clockwise direction whereas its anti clockwise away from the openings, generating an adequate fluid flow in all regions.

F. Temperature Vs Distance Plot

Chart Viewer is generated using temperature values of datum line at y = 0mm, y = 5mm, y = 10mm, y = 15mm, y = 20mm, y = 25mm, y = 30mm, y = 35mm, y = 40mm, y = 45mm and finally at y = 50mm. Along Y- axis Temperature is plotted and along X-axis distance from inlet is shown with origin at centre of heat sink. It is observed the lowest datum line shows the highest temperature (Bright red plot) due to its location near the source, and decreases as the datum height increases. The curve nature of the plot is same for all the data, concluding the heat distribution is uniform throughout.

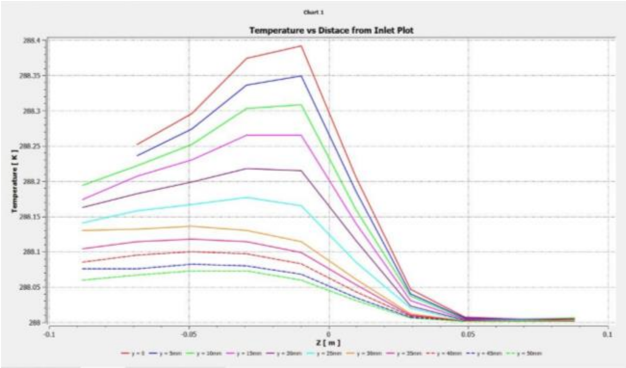


Figure 14. Temperature Vs Distance Plot

G. Pressure Vs Distance Plot

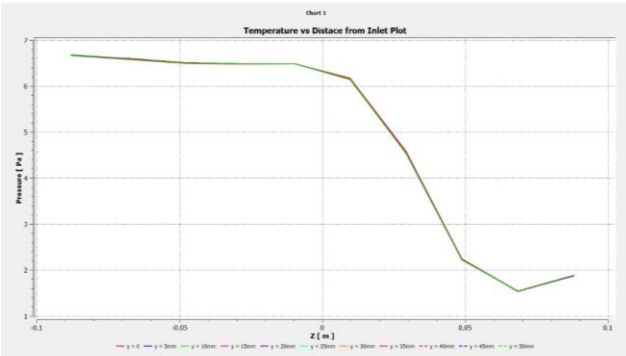


Figure 15. Pressure Vs Distance Plot

Chart Viewer is generated using Pressure values of datum line at y = 0mm, y = 5mm, y = 10mm, y = 15mm, y = 20mm, y = 25mm, y = 30mm, y = 35mm, y = 40mm, y = 45mm and finally at y = 50mm. Along Y- axis Pressure is plotted and along X-axis distance from inlet is shown with origin at centre of heat sink. No such observable pressure differences were spotted, concluding a uniform flow throughout.

VI. Conclusions

Concerned about the effective convective heat transfer from source via micro fins demanded a proper design of fins for better efficiency and adequate cooling of source. Using thin plate fins instead of cylindrical or cuboidal fins as heat sinks provides efficient forced convective heat transfer from the source to the cooling fluid circulated. In this article, a modified approach of effective fin design has been numerically simulated; the results have been analyzed in order to come to the following conclusions:

- Arranging the thin plate fins in rows and columns allows a proper arrangement for optimal fluid flow throughout resulting in proper heat transfer from all regions, cooling process is speeded up with proper and adequate flow.
- The source is positioned below the heat sink which being a good conductor of heat readily absorbs the excess heat from source via conduction, preventing excessive heating up of source, safeguarding the appliance from damage by heat.
- Increasing the thickness of base of heat sink increases the volume which in turn increase the heat absorbing capacity. The duration for heat transfer from source to heat sink up to maximum capacity is also increased many times which ensures proper safety of device.
- Inlet and Outlet areas are kept at maximum to induce the bulk amount of fluid flow in the heat sink region in optimal volume enhancing faster cooling of the device.
- The conclusions are exclusive of certain other optimizations which could not be achieved due to lack of data and computational applications. A number are other factors must be taken in consideration before considering the above results are promising. Those parameters and the simulations concerning those trails will take much more than just a numerical analysis and are left for further optimizations in future.

REFERENCES

[1] Danish Ansari, Kwang- Yong Kim, “Hotspot thermal management using a microchannel-pinfin hybrid heatsink,” *International Journal of Thermal Sciences,* vol. 134, pp. 27-39, December 2018.

[2] Craig E. Green, Peter A. Kottke, Thomas E. Sarvey, Andrei G. Fedorov, Yogendra Joshi, Muhannad S. Bakir, “Performance and Integration Implications of Addressing Localized Hotspots Through Two Approaches: Clustering of Micro Pin-Fins and Dedicated Microgap Coolers,” *ASME 2015 International Technical Conference and Exhibition on Packaging and Integration of Electronic and Photonic Microsystems collocated with the ASME 2015 13th International Conference on Nanochannels, Microchannels, and Minichannels,* November 19, 2015. https://doi.org/10.1115/IPACK2015-48537

[3] Soon, Noramah, “Numerical simulation of a microchip cooling with microjet array,” *Jurnal Mekanikal,* pp. 39-49, June 2008.

[4] S.E. Ghasemi, AA. Ranjbar, MJ. Hosseini, “Experimental evaluation of cooling performance of circular heat sinks for heat dissipation from electronic chips using nanofluid,” *Mechanics Research Communications,* vol. 84, pp. 85-89, September 2017.

[5] Bruno Agostini, MatteoFabbri, Jung E. Park, Lesjak Wojtan, John R. Thome, Bruno Michel, “State of the art of high heat flux cooling technologies,” *Heat Transfer Engineering,* vol. 28, pp. 258-281, March 2007. https://doi.org/10.1080/01457630601117799

[6] M.Rafati, A.A.Hamidi, M.Shariati Niaser, “Application of nanofluids in computer cooling systems (Heat transfer performance of nanofluids),” *Applied Thermal Engineering,* vol. 45-46, pp. 9-14, December 2012. https://doi.org/10.1016/j.applthermaleng.2012.03.028

[7] SS Gajghate, MA Patil, SR Babar, VC Ingavale, “Study of literature for different methods of cooling of microchip,” *IEEE International Conference on Emerging Trends in Engineering, Technology and Science ICETETS-2016.* February 2016.