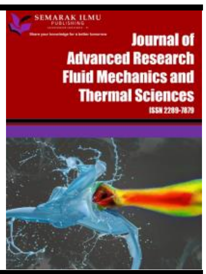
SEMARAK ILMU PUBLISHING

Journal of Advanced Research in Fluid Mechanics and Thermal Sciences

Journal homepage: https://semarakilmu.com.my/journals/index.php/fluid_mechanics_thermal_sciences/index
ISSN: 2289-7879



Numerical Study on Heat Propagation in Laptop Cooling System

Wong Mian Sheng[1], Abdulhafid M Elfaghi[1,*], Lukmon Owolabi Afolabi[1]

[1] Faculty of Mechanical and Manufacturing Engineering, Universiti Tun Hussein Onn Malaysia, Batu Pahat, 86400 Johor, Malaysia

ARTICLE INFO

Article history:
Received 8 April 2022
Received in revised form 20 July 2022
Accepted 30 July 2022
Available online 27 August 2022

Keywords:
Heat propagation; laptop cooling; CFD; heat sink

ABSTRACT

Thermal management of microelectronic devices has become increasingly important to maintain their performance and prolonging the lifespan of the devices. In this study, numerical simulation has been conducted to investigate the heat propagation for high performance microelectronic device of laptop. Two models are constructed and Ansys Fluent is used for the Computational Fluid Dynamics (CFD) simulation, source term is applied at the heat source, heat sink and two different material heat pipes are the main cooling components in the system. The results show that the improved design model is a better design for laptop cooling systems, and the increasing number of air vents, thermal conductivity, and length of heat pipes can effectively cool the high-powered microchip effectively. Transient simulation, considering the wick structure and working fluid in the simulation, is suggested for future work.

1. Introduction

Computational fluid dynamics (CFD) is an excellent tool for engineers and researchers. With the increasing of computing power and development of both physical models and numerical and discretization techniques, CFD is regarded as a very valuable tool for examining fluid flows [1-3].

Nowadays, high performance and low noise cooling system is one of the most important factors when consumer considering purchasing the electronic product such as laptop. Temperature of the operating machine is always an important topic in application ranging from laptop to turbo-engine in aircraft. An additional active laptop cooler uses small fans to generate additional air flow around the body of the laptop to conduct the heat away from the device [4]. Passive electronics cooling, such as a heat sink and heat pipe, can transfer the heat generated by a heat source component to a liquid medium, often air or a refrigerant where the heat is dissipated away from the device to allow or maintain the device's temperature at its designed operating temperature. There is a wide range of research done on thermal control devices, such as enhancing heat rejection with supercritical carbon carbide microchannel heat exchanger [5], shape memory alloy heat exchanger that improves the automatability of marching cooling capacity [6]. Structure design such as subcooled [7], two-phase stacked microchannel heat exchanger [8], working fluid, and materials [9] for the heat exchanger are

* *Corresponding author.*
E-mail address: abdulhafid@uthm.edu.my

https://doi.org/10.37934/arfmts.99.1.5865

always some popular topics for research. Working fluid for a cooler such as nanoparticle solution which had been used to enhance heat transfer [10], enhance thermal performance and uniformity [11], and higher heat transfer coefficient [12] in microelectronic devices cooling applications.

In recent development of these electronic products, most of them required a high-performance and low-noise cooling system to cool down the chip, such as central processing units (CPUs) or graphics processing units (GPUs) to maintain the performance of the machine. Temperature of chip increase when increasing usage of the machine. The performance of the chip will decrease as the temperature is too high. Thermal simulation can now be performed using fluid flow analysis such as Computational Fluid Dynamics (CFD). Engineers and researchers can test their design and concept without wasting more time and resources, then follow by validating their simulation results with experimental results to prove their concept. CFD simulation is used to perform thermal analysis on the cooling system such as loop heat pipe in the laptop [13] which shows a better cooling efficiency and low energy consumption. Embedded heat pipes and H-shape arrangement shows better cooling performance [14]. The fin of heat sinks that provides enhanced heat dissipation [15] and a passive cooling solution such as a graphene coating shows a better cooling performance than the active cooling solution in terms of heat dissipation, energy consumption, reliability, and noise-free operation [16].

Demand for high-performance cooling system in the market is increasing due to the continuous development of CPU chip, graphic card which require high-powered. To prolong the lifespan of electronic components, correct cooling strategy must be applied for the device to operate at optimal condition. Innovative thermal design is required for the recent development of miniaturized devices such as laptops to maintain their performance and portability. Few studies have focused on numerical thermal analysis of laptop cooling system. This aim of this study is to investigate the heat propagation in a microelectronic device, the CPU, and GPU inside a laptop using Computational Fluid Dynamics software.

2. Methodology

The governing equations for fluid flow and heat transfer are the following form of the incompressible continuity equation, the Navier-Stokes momentum equation, and the energy equation [17-19].

Continuity Equation

$$\nabla \cdot \vec{V} = 0 \tag{1}$$

Momentum Equation

$$\nabla^2 \cdot \vec{V} = 0 \tag{2}$$

Energy Equation

$$k\nabla^2 T = 0 \tag{3}$$

There are two models used in this study which Model 1 is the initial design while Model 2 is the improved design. The laptop chassis as shown in Figure 1 for Model 1 and Figure 2 for Model 2 are constructed in dimensions of H×W×D = 252×373×26 mm. The two main components that generate high heat flux, which is the aim of this study, are the CPU and the GPU. The heat sinks are modelled with dimension of H×W×D = 12×69×20 mm with 17 rectangular microchannels with the dimension H×W=10×3 mm. The inlet is located at the bottom for the chassis while the outlet is at the microchannels of heat sink. Model 2 meshing is illustrated in Figure 3.

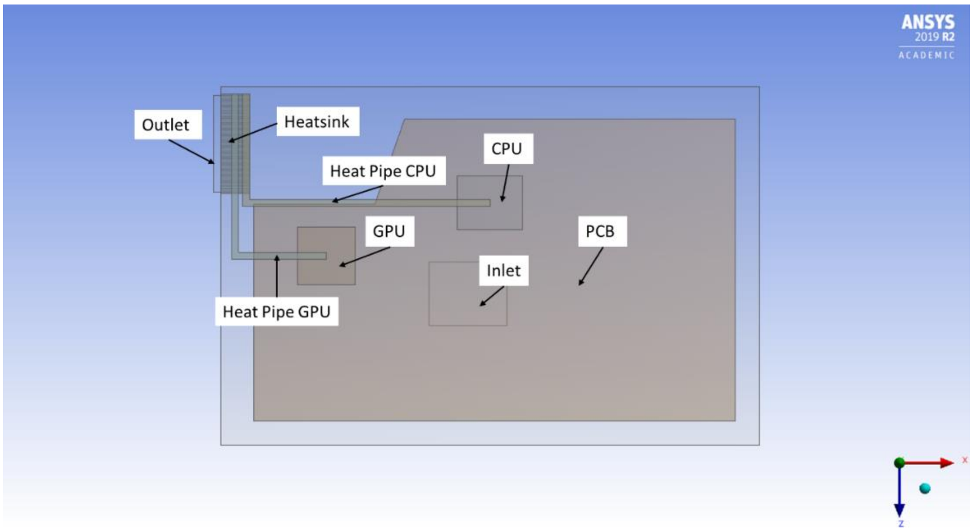


Fig. 1. Model 1

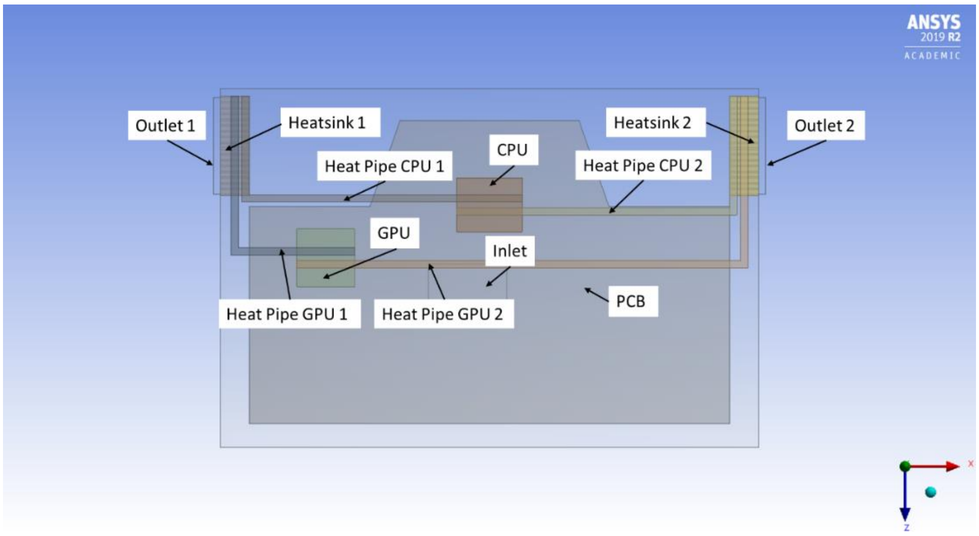


Fig. 2. Model 2

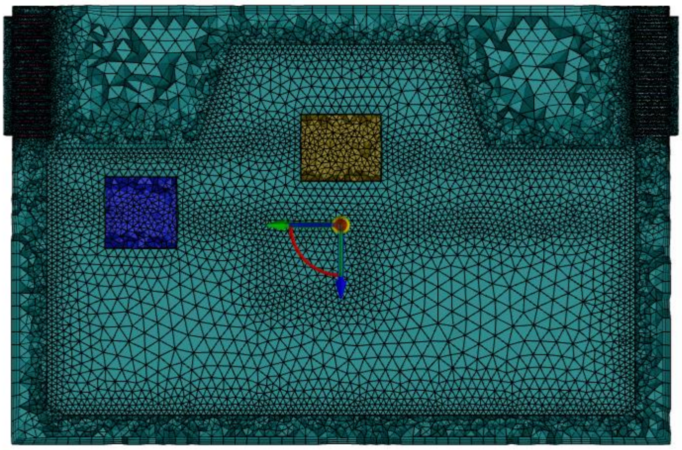


Fig. 3. Model 2 mesh

The specifications and boundary conditions for the simulation as shown in Table 1. The inlet, is defined as the pressure inlet while the outlet is defined as the mass flow outlet. The CPU and GPU power input is using the source term in Ansys Fluent.

Table 1
Specifications and boundary conditions

Components	Initial Conditions
Pressure Inlet	Atmospheric pressure
Mass Flow Outlet	0.001973 kg/s
CPU	10W, 90W
GPU	10W, 90W

The mass flow rate is calculated using the volume flow rate of the laptop cooling fan 0.1 m3 / min taken from Sun and Huai-liang [12], while the source term is calculated by referring to Velardo *et al.,* [19]. Material properties are important factors that contribute to the efficiency of microelectronic cooling systems. Table 2 shows the properties of the material used in this study.

Table 2
Material Properties

Material	Density (kg/m^3)	Thermal Conductivity (W/m.K)	Specific Heat (J/kg.K)
Air	1.185	0.0242	1007
Copper	8978	387.6	381
Copper Heat Pipe 1	8960	20000	385
Copper Heat Pipe 2	8960	40000	385
Epoxy	1400	0.35	1000
Silicon	2330	148	705

The two different values of thermal conductivity for copper heat pipes represent two different heat pipes in terms of working fluids and the structure of the wick [19].

3. Results and Discussion

The CFD simulation results in Figure 4 and Figure 5 show the temperature distribution for Model 1 and Model 2 respectively. The heat generated from the CPU and GPU is directed away by the heat pipes to the heat sink. The heat sink is then cooled by the laptop cooling fan. The CFD simulation

Journal of Advanced Research in Fluid Mechanics and Thermal Sciences
Volume 99, Issue 1 (2022) 58-65

results is similar to the previous studies results where the heat source has the highest temperature among other components and the power (watt) is also used to specify the heat generation by CPU and GPU with 10W and 90W by using source term in Ansys Fluent. Therefore, the working procedure and simulation results in this study can be validated.

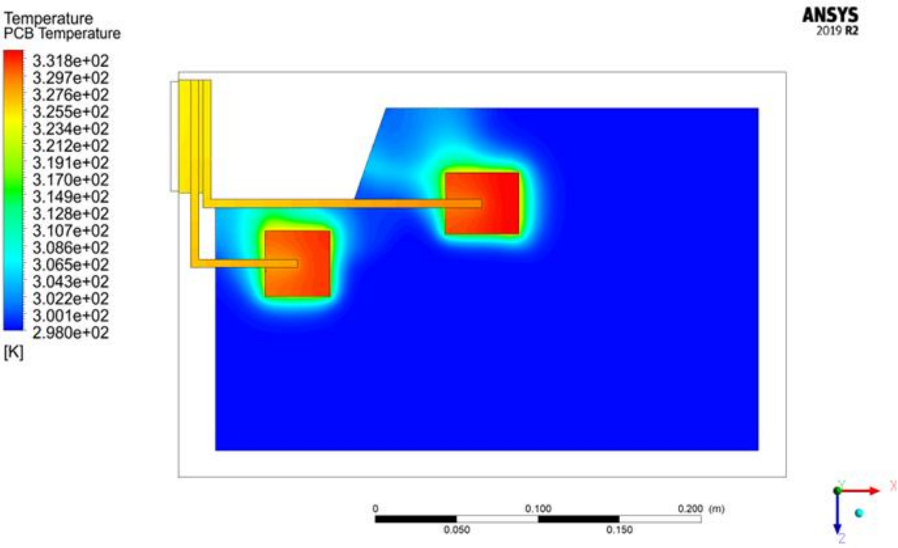


Fig. 4. Simulation Result for model 1 with 10W at CPU and GPU

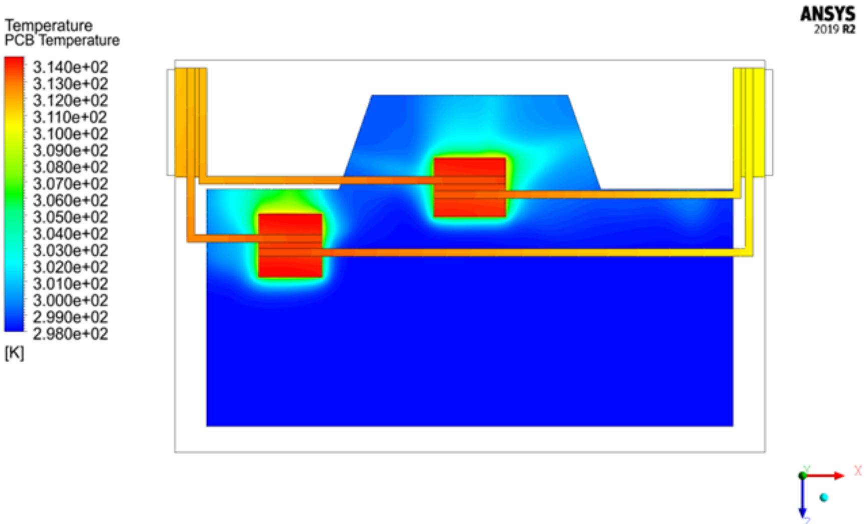


Fig. 5. Simulation result for model 2 with 10W at CPU and GPU

The temperature contour in Figure 6 shows that Model 2 has better cooling than Model 1 for both 10W and 90W power supplied to CPU and GPU. The average temperature for CPU and GPU has a significant reduce in Model 2 compared to Model 1. Increasing the number of heat pipes provides enhancement in cooling efficiency and reduces the overall temperature for laptop chassis. This is due to the heat being conducted away by more heat pipes. Increasing the thermal conductivity of heat pipes and the air vent also contributes to improve the cooling efficiency for the system. The longer heat pipe provides a greater temperature reduction at the heat source compared to the shorter heat pipe. As a result, the CPU in Model 1 and GPU in Model 2 shows a greater temperature drop when increasing the thermal conductivity of heat pipes. The temperature at heat source will also affect the cooling efficiency heat pipes. The higher the temperature at the heat source, the more temperature reduce when increasing thermal conductivity for the heat pipes. Furthermore, the total heat transfer rate in the contact area between the CPU and the GPU with the heat pipe for Model 2 is also higher than that of Model 1. The value for total heat transfer rate is as listed in Table 3.

Table 3
Total heat transfer rate

Cases	CPU and Heat Pipe (W)	GPU and Heat Pipe (W)
Model 1 90W Power with CHP1	67.260	68.714
Model 1 90W Power with CHP2	68.279	69.300
Model 2 90W Power with CHP1	75.220	78.072
Model 2 90W Power with CHP2	76.056	78.746

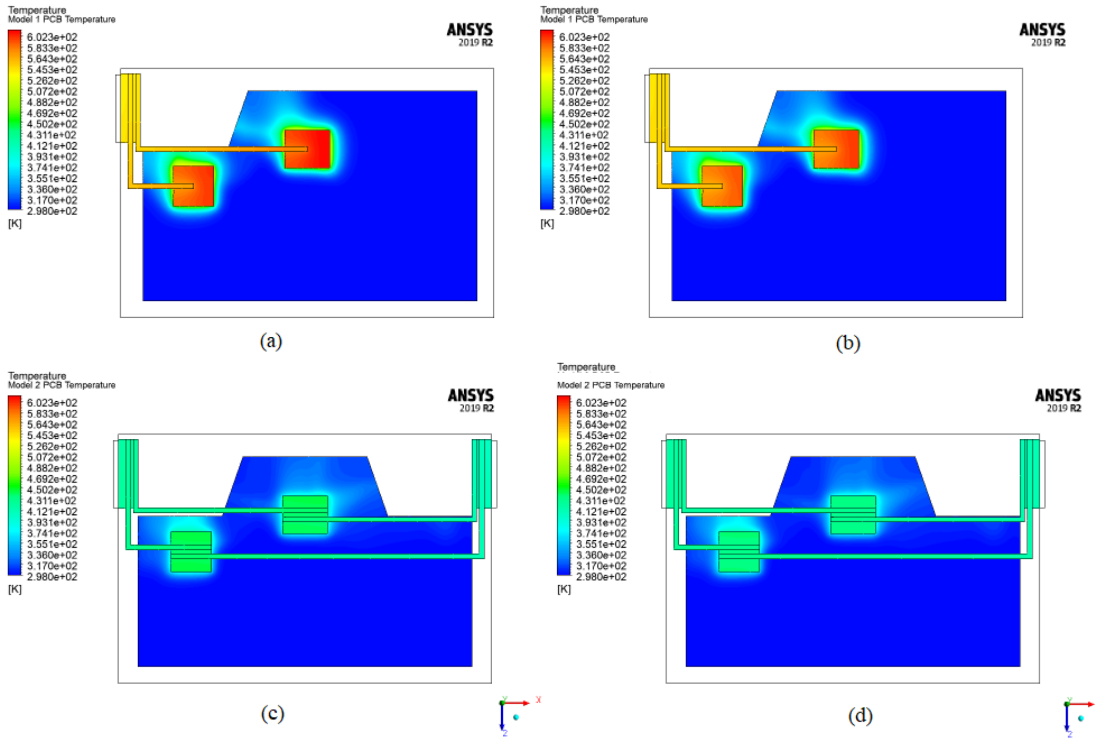


Fig. 6. Temperature contour for 90W power (a) Model 1, CHP1, (b) Model 1, CHP2, (c) Model 2, CHP1, (d) Model 2, CHP2

Based on Figure 7, the velocity of air flow in Model 2 is higher than in Model 1. This is due to Model 2 have two outlets and mass flow rate for laptop cooling fans is applied at both outlets. Increasing the number of laptop cooling fan and air vent for laptop chassis can improve the cooling rates. Also, increasing the fan speed can further speed up the air circulation inside the laptop chassis and exhaust the hot air faster.

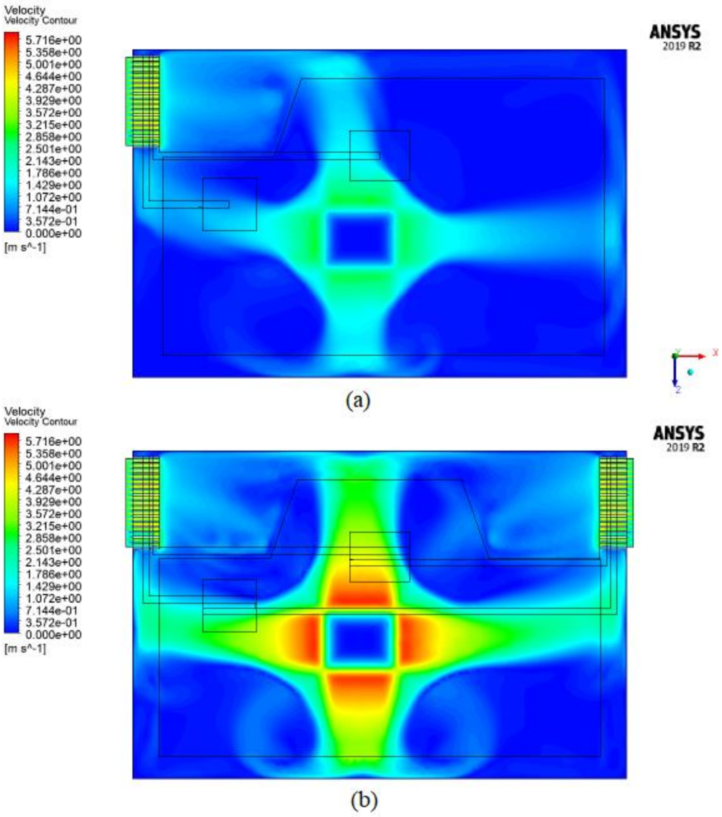


Fig. 7. Velocity Contour (a) Model 1, (b) Model 2

4. Conclusions

In this study, CFD simulation on laptop cooling system was presented. Two simplified laptop chassis were modelled, consisting of two main heat sources CPU and GPU, heat pipes, heat sink, and PCB. The heat propagation for microelectronic devices of laptop is analysed. The proposed cooling system design which is the Model 2 shows a better cooling than Model 1 for both 10W and 90W power. However, Model 1 is more suitable to be used for 10W power since it is already capable enough to maintain the operating temperature for CPU and GPU. Model 2 shows a significant improvement in 90W power as the temperature for CPU and GPU substantially reduces due to the increase in number of heat pipes, air vent, and laptop cooling fan. In addition, increasing the thermal conductivity and length of the heat pipes can also improve the cooling rates for the system. Further recommendation on this topic to improves the results is needed such as perform transient simulation, vary the power supplied according to the usage can obtain a more accurate real life device usage result.

Acknowledgement
The authors acknowledge the Research Management Centre, (RMC), Universiti Tun Hussein Onn Malaysia for the financial support through TIER 1 research grant (H768).

References

[1] Abobaker, Mostafa, Sogair Addeep, Lukmon O. Afolabi, and Abdulhafid M. Elfaghi. "Effect of Mesh Type on Numerical Computation of Aerodynamic Coefficients of NACA 0012 Airfoil." *Journal of Advanced Research in Fluid Mechanics and Thermal Sciences* 87, no. 3 (2021): 31-39. https://doi.org/10.37934/arfmts.87.3.3139

[2] Elfaghi, Abdulhafid M., Wisam B. Ajaj, and Lukmon Owolabi Afolabi. "Effect of Oil Mass Flow Rate on Temperature Profile in Oil Wells." *Journal of Advanced Research in Fluid Mechanics and Thermal Sciences* 77, no. 2 (2021): 23-32. https://doi.org/10.37934/arfmts.77.2.2332

[3] Afolabi, Lukmon Owolabi, Oluwafunke T. Afolabi-Owolabi, Abdulhafid M. Elfaghi, Djamal Hissein Didane, Mohammed Ghaleb Awadh, and Al-Mahmodi Akram. "Thermal Characterization of Biofluids for Heat Transfer Fluid in Thermal Transport Technologies." *Journal of Advanced Research in Fluid Mechanics and Thermal Sciences* 89, no. 1 (2022): 134-141. https://doi.org/10.37934/arfmts.89.1.134141

[4] Zheng, Qiye, Menglong Hao, Ruijiao Miao, Joseph Russell Heinlen Schaadt and Chris Dames. "Advances in thermal conductivity for energy applications: a review." *Progress in Energy* 3 (2020): 012002. https://doi: 10.1088/2516-1083/abd082

[5] Olivia C. da Rosa, Felipe G. Battisti, Gustavo M. Hobold, Alexandre K. da Silva. "Enhancing heat rejection from electronic devices with a supercritical carbon dioxide minichannel heat exchanger." *International Journal of Refrigeration* 106 (2019): 463-473. https://doi.org/10.1016/j.ijrefrig.2019.07.008

[6] Chu, Xuyang, Huihui You, Xiao Ling Tang, Wei Zhou, Xinying Li, Ding Yuan and Shupan Zhou. "Smart microchannel heat exchanger based on the adaptive deformation effect of shape memory alloys." *Energy Conversion and Management* 250 (2021): 114910 https://doi.org/10.1016/j.enconman.2021.114910

[7] Ariyo, David O. and Tunde Bello-Ochende. "Constructal design of subcooled microchannel heat exchangers." *International Journal of Heat and Mass Transfer* 146 (2020): 118835. https://doi.org/10.1016/j.ijheatmasstransfer.2019.118835

[8] Ariyo, David O. and Tunde Bello-Ochende. "Constructal design of two-phase stacked microchannel heat exchangers for cooling at high heat flux." *International Communications in Heat and Mass Transfer* 125 (2021): 105294. https://doi.org/10.1016/j.icheatmasstransfer.2021.105294

[9] Ali, Saad, Faiz Ahmad, Puteri S. M. Megat Yusoff, Norhamidi Muhamad, Eugenio Oñate, Muhammad Rafi Raza and Khurshid Malik. "A review of graphene reinforced Cu matrix composites for thermal management of smart electronics." *Composites Part A-applied Science and Manufacturing* 144 (2021): 106357. https://doi.org/10.1016/j.compositesa.2021.106357

[10] Al-Rjoub, Marwan F., Ajit K. Roy, Sabyasachi Ganguli and Rupak Kumar Banerjee. "Enhanced heat transfer in a micro-scale heat exchanger using nano-particle laden electro-osmotic flow." *International Communications in Heat and Mass Transfer* 68 (2015): 228-235. https://doi.org/10.1016/j.icheatmasstransfer.2015.09.009

[11] Maganti, Lakshmi Sirisha, Purbarun Dhar, Thirumalachari Sundararajan and Sarit K. Das. "Mitigating non-uniform heat generation induced hot spot(s) in multicore processors using nanofluids in parallel microchannels." *International Journal of Thermal Sciences* 125 (2018): 185-196. https://doi.org/10.1016/j.ijthermalsci.2017.11.015

[12] Sun, Bin and Huai-liang Liu. "Flow and heat transfer characteristics of nanofluids in a liquid-cooled CPU heat radiator." *Applied Thermal Engineering* 115 (2017): 435-443. https://doi.org/10.1016/j.applthermaleng.2016.12.108

[13] Wang, Yabo, Bin Wang, Kai Zhu, Hailong Li, Wei He and Shengchun Liu. "Energy saving potential of using heat pipes for CPU cooling." *Applied Thermal Engineering* 143 (2018): 630-638. https://doi.org/10.1016/j.applthermaleng.2018.07.132

[14] P. Bishnoi, M. Srivastava, and M. Sinha, "CFD ANALYSIS OF CPU FOR COOLING OF DESKTOP COMPUTERS," *International Journal of Advanced Technology in Engineering and Science* 4 (2016): 693-700.

[15] Al-Baghdadi, Maher. "Graphene-Coating for Efficient Electronics Cooling." (2020). https://doi.org/10.32545/encyclopedia202007.0012.v1

[16] Elfaghi, Abdulhafid M., Ashraf Ali Omar and Waqar Asrar. "Higher-Order Compact-Flow Field-Dependent Variation (HOC-FDV) Method for Solving Two-Dimensional Navier-Stokes Equations." *International Journal for Computational Methods in Engineering Science and Mechanics* 16 (2015): 256 - 263. https://doi.org/10.1080/15502287.2015.1048386

[17] Ennil, Ali S. Bahr, and Abdulhafid M. Elfaghi. "Numerical simulation of film cooling over flat plate." *New Approaches in Engineering Research Vol. 12* (2021): 10-16. https://doi.org/10.9734/bpi/naer/v12/8210D

[18] Abobaker, Mostafa, Abdulhafid M. Elfaghi, and Sogair Addeep. "Numerical Study of Wind-Tunnel Wall Effects on Lift and Drag Characteristics of NACA 0012 Airfoil." *CFD Letters* 12, no. 11 (2020): 72-82. https://doi.org/10.37934/cfdl.12.11.7282

[19] Velardo, Jason, Randeep Singh, Ashwin Date and Abhijit Date. "An Investigation into the Effective Thermal Conductivity of Vapour Chamber Heat Spreaders." *Energy Procedia* 110 (2017): 256-261. https://doi.org/10.1016/j.egypro.2017.03.136