Jurnal Kejuruteraan 31(1) 2019: 139-147
https://doi.org/10.17576/jkukm-2019-31(1)-17

Computational Fluid Dynamics Simulation on the Heat Sink Performance of a Graphics Processing Unit Thermal Management

(Simulasi Pengkomputeran Dinamik Bendalir terhadap Penyerapan Haba bagi Pengurusan Terma Unit Pemprosesan Grafik)

Zambri Harun*, Ng Jie Suang, W. M. Faizal W. Mahmood, Mahir Faris Abdullah & Eslam Reda Lotfy
Centre for Integrated Design for Most Advanced Mechanical System, Universiti Kebangsaan Malaysia, Malaysia
**Corresponding author: zambri@ukm.edu.my*

Received 11 September 2017, Received in revised form 1 July 2018
Accepted 28 January 2019, Available online 30 April 2019

ABSTRACT

The article focuses on the numerical investigation of temperature distribution in a central processing units (CPU) case with different time interval such as t = 100 s, 200 s, 300 s, 400 s, 500 s. Heat sink performance of a graphics processing unit (GPU) thermal management and impacts of different shape and velocity on the thermal performance are considered. In this study, three heat sink models are designed (A, B, and C) based on the volume area of heat sink. This study emphasizes the heat transfer phenomena caused by a GPU in a computer case in both steady state and transient state. A CFD software STAR – CCM + is used to carry out to study the fluid flow and heat transfer simulation of graphics card heat sink in a computer case and the same time an enhanced method of reducing the temperature of GPU is proposed. The results show that heat sink B with the least volume area, has the fastest rate of heat exchange followed by heat sink C and heat sink A. Likewise, the result indicates an inverse relationship between the volume and the total surface of the heat sink and the final temperature of the graphics card chip. As the total volume and surface of the heat sink increases, the rate of heat transfer increases via faster rate of conduction between graphics card chip to heat sink meanwhile the cooling of the heat sink is aided by wind inlet via convection.

Keywords: Heat Sink; Heat Transfer; Steady State; Transient State

ABSTRAK

Artikel berfokus kepada penyiasatan berangka pengedaran suhu dalam kes CPU dengan selang masa yang berlainan iaitu t = 100 s, 200 s, 300 s, 400 s, 500 s. Prestasi haba sinki unit pengurusan pemproses grafik (GPU) dan kesan bentuk dan halaju yang berlainan pada prestasi terma dipertimbangkan. Dalam kajian ini, tiga model haba terbenam direka (A, B, dan C) berdasarkan luas keluasan sink haba. Kajian ini menekankan fenomena pemindahan haba yang disebabkan oleh GPU dalam kes komputer dalam keadaan mantap dan keadaan sementara. Perisian CFD STAR – CCM + digunakan untuk mengkaji aliran cecair dan simulasi pemindahan haba kad grafik haba sink dalam kes komputer dan pada masa yang sama kaedah yang dipertingkatkan untuk mengurangkan suhu GPU dicadangkan. Keputusan menunjukkan bahawa sink haba B dengan jumlah luas isipadu minimum, mempunyai kadar pertukaran haba terpantas diikuti oleh sink haba C dan sink haba A. Begitu juga, hasil menunjukkan hubungan songsang antara jumlah dan permukaan keseluruhan sink haba dan suhu akhir kad cip grafik. Memandangkan jumlah isipadu dan permukaan sink haba meningkat, kadar pemindahan haba meningkat melalui kadar pengaliran lebih cepat antara cip kad grafik ke haba sinki sementara penyejukan sink haba dibantu oleh salur angin melalui perolakan.

Kata kunci: Haba Terbenam; Pertukaran Haba; Keadaan Mantap; Keadaan Sementara

INTRODUCTION

The study on heat transfer has attracted the attention of many researchers because of its significant impact in increasing the efficiency of engineering and industrial applications (Chai et al. 2013; Abdullah et al. 2017; Yu et al. 2014). One of the applications is a heat sink performance of a graphic processing unit (GPU) thermal management and impact of it on the thermal performance (Xia et al. 2011, 2013). GPUs in high-performing graphic computing become heat source. As temperature increases, the performance of the graphics card drops causing the same to graphic performance and in an extreme case, causing a computer operations to stop. A heat sink is used to overcome the overheating issue of the graphics card. However, the transfer and flow of heat inside a computer case and distribution of temperature at the heat sink are unknown. Hence, a computational fluid dynamics (CFD) simulation is carried out to investigate and observe the transfer of heat flow inside a computer case and heat sink in order to solve the graphics card overheating issue.

Some research on heat sink cooling has been carried out to enhance the thermal performance for the many applications (Liu et al. 2011; Kuppusamy et al. 2014) for example, using heat sink as a cooling device for components which dissipate large amounts of heat (Ferrari 1977). Based on this research, it is related to cooling of power thyristors by using a heat sink applying conduction and convection concept to prevent overheating of the power thyristors. Convection and conduction cooling concept from the research is almost similar to graphics card cooling system. Conduction occurrs when an electrical component transfers heat generated to the heat sink by surface to surface contact whereas convection occurs when a forced flow liquid coolant into and out of the heat sink which comprises of the inlet and outlet ducts. Martin (1995) invented a multimode heat sink for an electronic module. The rig was mounted within an electronic module mounting rack, and more particularly such that cooling could occur by conduction or convection. Lee (2003) also performed research to cool a chipset by using a heat sink device. The heat sink device for cooling of chipset mounted on a printed circuit board to interface a central processing unit (CPU) for dissipating heat generated by the chipset via convection process.

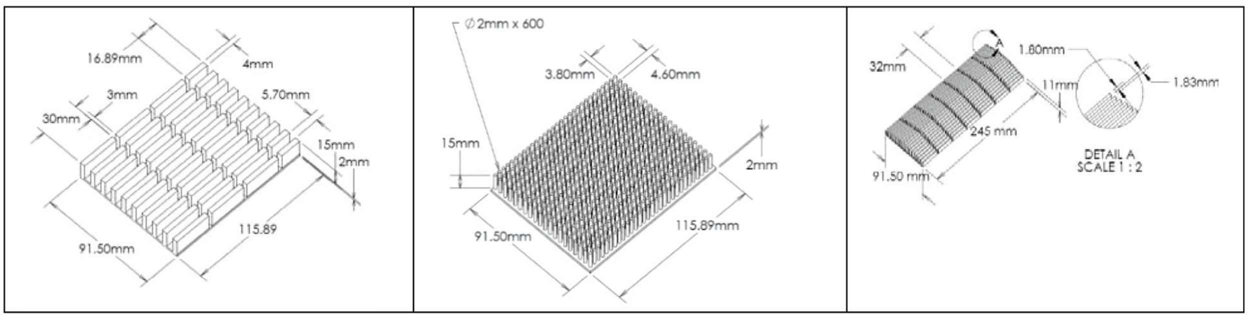
METHODOLOGY

A simplified graphics card inside a computer case is modeled to represent a real computer operation. The inlet and outlet region of the computer case is modeled like a normal computer case. The size, geometry, and material of the graphics card and heat sink are modeled similarly according to the graphics card available in the market.

STAR – CCM + software is used to perform the simulation. STAR – CCM + is a product of CD Adapco (computational dynamics-analysis & design application company Ltd.), is a multinational computer software company that authors and distributes application used for computer-aided engineering. STAR – CCM + allows users to solve problems from a lightweight computer and substantially reduces the need for the expensive remote machine.

GEOMETRY CONFIGURATION

A computer case, as shown in Figure 1, graphics cards are drawn by using Solidworks and converted into IGES file. The drawings were then imported into the STAR – CCM + software. The size of the computer case is 0.5 m × 0.5 m × 0.2 m. The size of the GPU is 30 cm × 10 cm × 2 cm.



(a) Heat sink A (b) Heat sink B (c) Heat sink C

FIGURE 1. Three heat sink case studies

In this project, three heats sink models are designed as shown in Figure 1(a), (b) and (c) to study the effect of geometry, total volume and surface area on the efficiency of heat transfer. The objective to design the heat sink in a different geometry, total volume, and surface area is to determine the distribution of heat contour on the heat sink to find out and solve the overheating of graphics card chip issue related to these factors. All the heat sinks are made up of the same material. The material of heat sink and the graphics card component will be discussed in the next part.

BOUNDARY CONDITIONS

Three-dimensional gradients of the model are selected for the overall domain of the computer case as the simulation will be carried out in 3D mode. Figure 2 shows the boundary conditions of the system consist of three parts, which are the graphics card, air and heat sink. The air flow consists of the air inlet and pressure outlet which have wind velocity of 1 ms^{-1} respectively. The air is assumed as steady flow and incompressible throughout the flow, and the air density is constant, 1.225 kg/m^3. The temperature of the air is assumed as ideal room temperature, 2525^{o}C. The pressure of air is assumed as ideal atmospheric pressure, 101325 Pa. The type of air flow is modeled as asegregated flow which is suit for incompressible flow,andfewer resources are required. The air is modeled as turbulent as the Reynolds number achieves turbulent criteria for internal flow (Re > 4000). Next, air is modeled into Reynolds-averaged Navier-Stokes because this is an equation for time-averaged equation of motion for fluid flow and primarily used while dealing with turbulent flow. *k-ε* turbulence is modelled for the air as it is the most common model to simulate mean flow characteristic for turbulent flow. Besides, it is simple to implement, with stable calculations, and converge easily.

Regarding the graphics card, the time is modeled as steady as the heat released by the GPU chip is considered constant flow at any point. The material of the graphic card is modeled to multi-part solid as the graphics card consists of many solid components. Segregated solid energy flow is used here as it suits constant density and incompressible flow and less resource are needed.

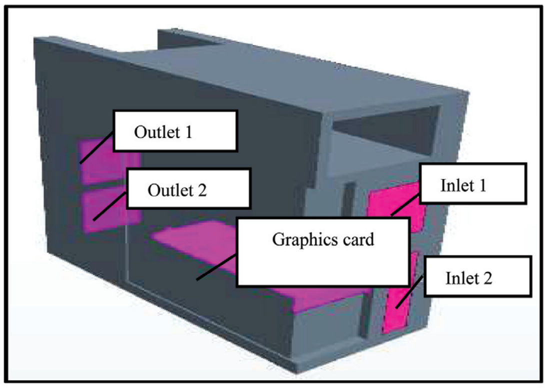


FIGURE 2. Boundary conditions

Finally, for the heat sink, the time is modeled as steady since the heat source generated is constant throughout the heat sink. The material of the heat sink is a model is solid as it only consists of a single type of material. Segregated flow is used to model the flow here due to fewer resources are needed. Figure 3 shows the internal part and the material of the GPU. The material properties of the component are according to the standard mechanical properties of each material. The material for heat sink and graphic card are shown in Table 1, all of the components are the heat source supplied to the system, and this will be further discussed in the physical model part.

TABLE 1. Material for each component

Component	Material
Heat sink	Aluminum
Capacitor	Alumina
Memory chip	Silicon
GPU chip	Aluminum
Printed circuit board (PCB)	FR-4 + copper
Connector port block	Aluminum
Back plate	Aluminum

MESHING

Figure 4 shows the meshing of the whole system, GPU heat sink and internal structure of GPU. The cells are more concentrated on the critical area such as heat sink. The type of mesh used is polyhedral meshing, and the minimum and maximum cell sizes are 2 mm and 10 mm respectively. The meshing generates 200,000 cells, 1,291,476 faces, 1,332,162 vertices. This already fulfills grid independence test, and the result of the simulation is converged when using this number of cells that as shown in Table 2.

TABLE 2. Grid independence test

Number of cells	Maximum temperature (Heat sink A with 1ms^{-1})
~50 k	110.52°C
~100 k	99.52°C
~200 k	90.52°C
~300 k	90.30°C

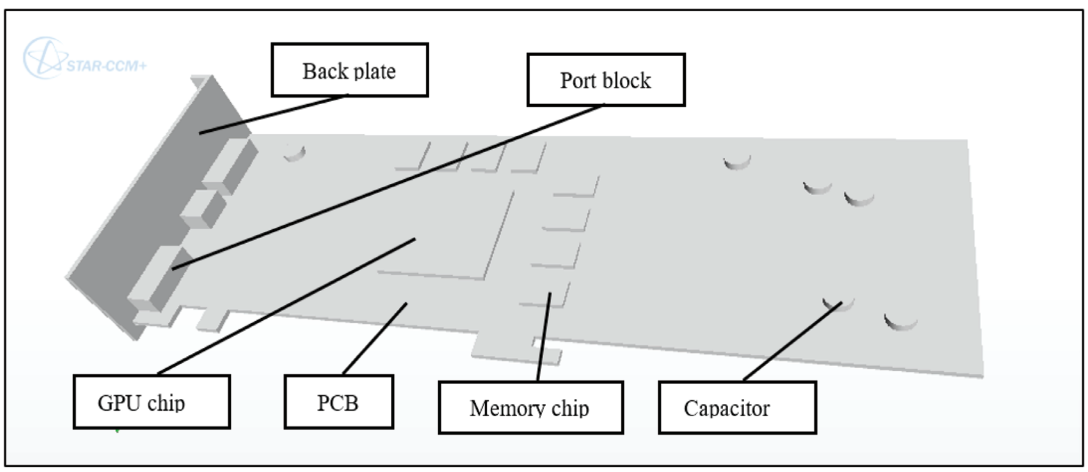


FIGURE 3. GPU internal part and material

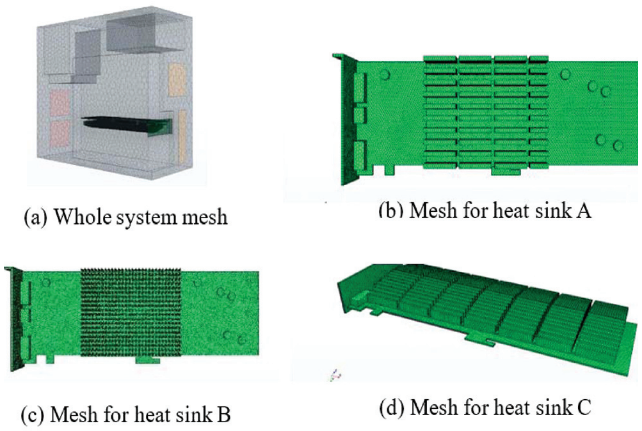


FIGURE 4. Meshing for whole system, GPU heat sink and internal structure of GPU

MATHEMATICAL MODELLING

GOVERNING EQUATIONS

It is often called the equation of continuity because it requires no assumptions except that the density and velocity are continuum functions. That is, the flow may be either steady or unsteady, viscous or frictionless, compressible or incompressible. The equation of continuity is:

$$\frac{\delta\rho}{\delta t}+\frac{\delta(\rho u)}{\delta x}+\frac{\delta(\rho v)}{\delta v}+\frac{\delta(\rho w)}{\delta w}=0 \quad (1)$$

Since the air flow is less than 0.3 of the speed of sound in air, the density of air is considered as constant.

This equation is valid for a Newtonian fluid under very general conditions of unsteady, compressible, viscous, heat- conducting flow, except that it neglects radiation heat transfer and internal sources of heat that might occur during a chemical or nuclear reaction.

$$\rho\frac{\delta\hat{u}}{\delta t}+\rho(\nabla.V)=\nabla.(k\nabla T)+\phi \quad (2)$$

REYNOLDS NUMBER

Separation of boundary layer depends on Reynolds number. For higher values of Reynolds number, it exhibits an early transition from laminar to turbulent flow. The higher the Reynolds number, there will be a greater tendency that the flow is turbulent.

Reynolds number is *Re* defined by Incropera et al. 2007,

$$R_e=\frac{\rho Vx}{\mu} \quad (3)$$

where x is the characteristic length of the airfoil (chord length of GPU).

TURBULENT MODEL

Turbulent model satisfies certain mathematical constraints on the Reynolds stresses, consistent with the physics of turbulent flows. The turbulence is modeled by the two-equation $k-\varepsilon$ model, this model is characterized as robust and more accurately predicts results in such application. Also, this model is likely to provide superior performance for flows involving rotation, boundary layers under strong adverse pressure gradients, separation, and recirculation. The most widely-used engineering turbulence model for industrial applications, which is easy and suitable to implement. Thus, the turbulence of flow field is expressed in turbulence kinetic energy (k) and dissipation rate (ε), using following equations (Bergman et al. 2011):

$$\frac{\delta}{\delta t}(\rho k)+div(\rho uk)=div(\Gamma_k\nabla k)+G-\rho\varepsilon \quad (4)$$

$$\frac{\delta}{\delta t}(\rho\varepsilon)+div(\rho u\varepsilon)=div(\Gamma_k\nabla\varepsilon)+C_1\frac{\varepsilon}{k}G \quad (5)$$

$$-C_2\rho\frac{\varepsilon^2}{k}$$

$$\mu_1=\rho C_\mu\frac{k^2}{\varepsilon}$$

$$\mu_{eff}=\mu+\mu_1$$

Where Γ, μ, σ, t are diffusion coefficients for k and ε, respectively, G is the production rate by turbulence shear stresses, μ is the molecular viscosity, and u is the turbulence viscosity. The terms $C1$, $C2$ are the turbulence Prandtl number.

HEAT TRANSFER

In this simulation, the heat transfer occurs via conduction and convection modes. There are three main regions emphasized in the simulation, namely air, GPU chip, heat sink. Conduction occurs in GPU chip to heat sink due to the GPU chip and the other component in contact with the heat sink (Yunus 2006). Convection occurs inside CPU case due to the heat transfer from the heat sink to the air inside CPU case via air inlet and outlet.

RESULT AND DISCUSSION

STEADY STATE

Figure 5 shows the contour of maximum temperature distribution on heat sink when steady state is reached for velocity inlet 1 ms^{-1} and 2 ms^{-1}.

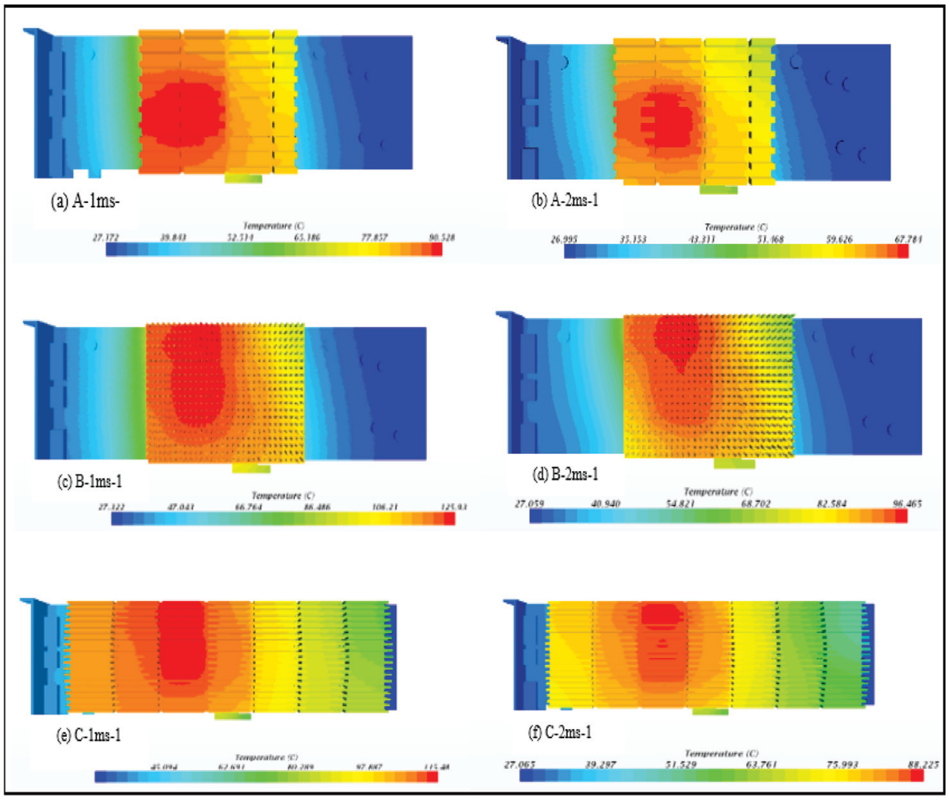


FIGURE 5. (a-f) Temperature contours on heat sink

In Figure 5, the highest temperature distribution occurs in the central part of the heat sink, and further from the central part of the heat sink, the temperature gets lower. Then the heat source spread slowly to another part of the heat sink. Here, there is a contact area between GPU chip and heat sink, causing conduction to occur. Conduction occurs when heat is released by the GPU chip and another component of electronic to the heat sink. After that, convection occurs when the heat in the heat sink is removed by the convecting air inlet. Fluid flow is multiphase here. Besides, this explains the reason electronic components should be arranged away or shielded from heat sources to prevent overheating and eventually causing damage. Furthermore, the maximum temperature of the heat sink is inversely proportional to the inlet wind. The higher the velocity of the air inlet causes the lower the maximum temperature of the heat sink, as shown in Table 3. This is because the velocity of the wind will speed up the rate the removal of heat from the heat sink and subsequently causes the convection process occurs faster to lower the maximum temperature (Bejan 2013). On the other hand, heat sink A has the lower maximum temperature because it has a maximum volume of area among three heat sinks. The higher the volume area, the higher the rate of heat loss (Leach 2005). Based on Table 3, the higher the volume area of the heat sink, the higher the rate of heat released via convection, the lower the maximum heat sink temperature when it reaches steady state. The thermal resistance of the pin-fin heat sink decreases with the Reynolds number. Appropriate placement of the vortex generators improves the heat transfer efficiency of the heat sink, and the effect is more significant at a lower Reynolds number (*Re* = 5000) (Li et al. 2017).

TABLE 3. Maximum temperature of heat sink for different velocity

Heat sink (ms^{-1})	Velocity inlet temperature (ºC)	Maximum (mm^3)	Volume area
A	1	90.52	6.31
	2	67.78	6.31
B	1	125.94	4.03
	2	96.467	4.03
C	1	115.48	4.3
	2	88.225	4.3

The conduction processes in the graphics card cooling system is similar to the one in the car radiator. A car radiator uses a huge number of fins as a way to increase heat transfer rate to cool down the water temperature. Figure 6 shows the relationship between the volume area (number of fins) and the rate of heat transfer. The same concept as the graphic heat sink cooling system here, the higher volume of the area, the higher the rate of heat transfer, causing the water temperature in a car can be cool down faster (Shah 2003).

For convection, the rate of heat transfer via convection highly depends on the air velocity. The coefficient of heat convection is proportional to the air velocity. The higher the air velocity, the higher the heat coefficient convection, and the higher the rate of heat transfer (Khabari et al. 2014).

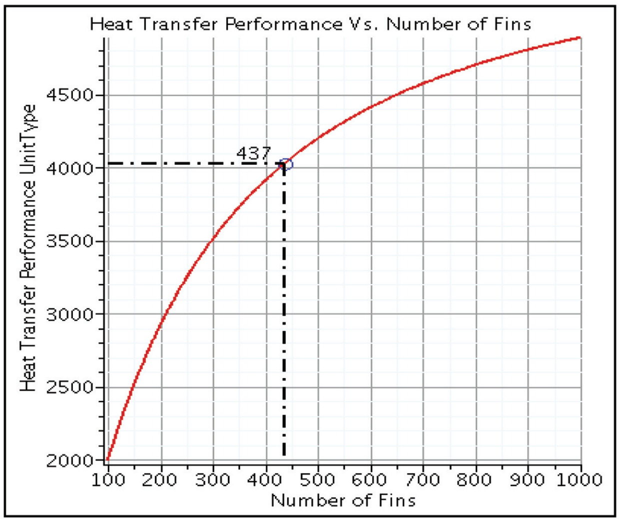


FIGURE 6. Heat transfer performance against number of fins: (Maplesoft et al. 2008)

Figure 7 shows a graph of the relationship between convective heat transfer coefficient against air velocity (Khabari, A., M. Zenouzi 2014). The same concept with the graphics cooling system here, the higher the velocity inlet of wind can increase the rate of convection, causing the heat transfer to occur at a high rate to speed up the removal of heat from the heat sink.

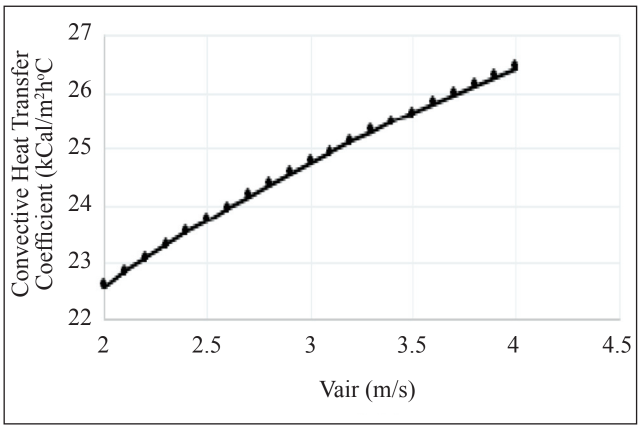


FIGURE 7. Graph of relationship between convective heat transfer coefficients against air velocity: (Khabari et al. 2014)

Figure 8 shows the maximum temperature of GPU chip when it reaches steady state for velocity inlet 2 ms^{-1}. Based on Figure 8, the temperature of the GPU chip increases with the time interval until it becomes stable. The reason why the temperature is different has been discussed in the previous section. In the simulation, assumption is made such that the graphic card is used violently (performing simulation, rendering for 4k ultra gaming). Hence, it will have a higher temperature than normal.

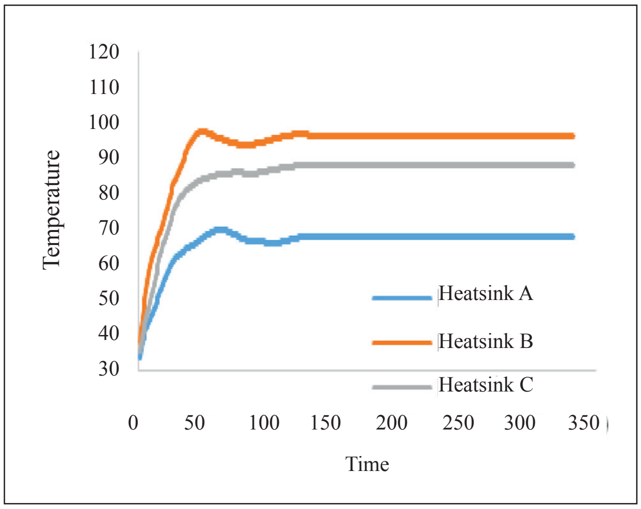


FIGURE 8. Maximum temperature for heat sink A, B, C at steady state

TRANSIENT MODES

A system is said to be in a transient state when a process variable changes and the system has not yet reached steady-state conditions (Smith, 1975). Figure 9 shows the comparison of temperature distribution contour of each heat sink for speed 1 ms^{-1} at a different time interval (t = 100 s, 200 s, 300 s, 400 s, 500 s).

From time interval t = 100 s until 500 s, as shown in Figures 9, the heat started to spread to the heat sink slowly for all heat sink cases. Based on the contour temperature distribution, heat sink B has the fastest rate of heat exchange followed by heat sink C and heat sink A.

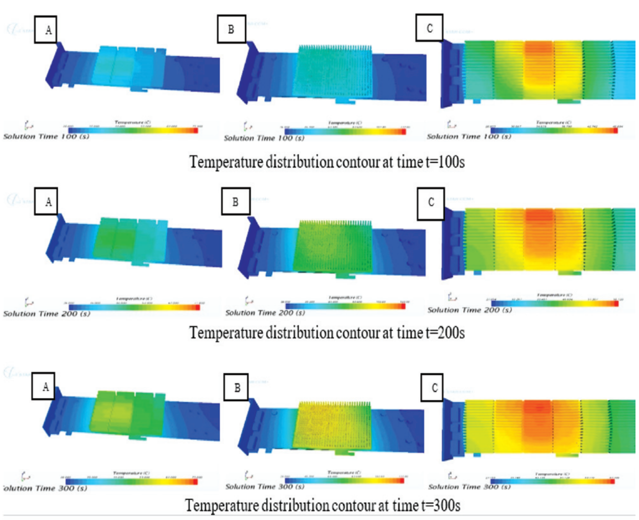


FIGURE 9.A. Temperature distribution contour of each heat sink for speed 1 ms^{-1} at different time interval

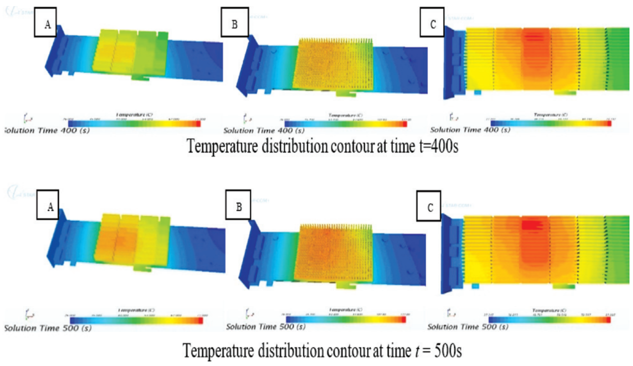


FIGURE 9.B. Temperature distribution contour of each heat sink for speed 1 ms^{-1} at different time interval

From time interval t = 100 s until 500 s, as shown in Figures 9, the heat started to spread to the heat sink slowly for all heat sink cases. Based on the contour temperature distribution, heat sink B has the fastest rate of heat exchange followed by heat sink C and heat sink A. Heat sink B experiences highest heat exchange due to heat sink B has the smallest volume area which has already been discussed in the previous section, Table 3. Furthermore, according to the thermodynamics concept, , where, is heat transfer rate per unit area, is the amount of heat transfer (Abdullah et al. 2017), is the time interval, is area (Bolgarskii 1967). Based on the law, it shows that the rate of heat transfer is inversely proportional to the area. Same concept is applied at here. At the beginning, the temperature at the initial condition is the same i.e. 26ºC, with the increase of time interval, the temperature for all heat sink increase gradually due to conduction between the GPU chip (heat source) and the heat sink. At the final stage, heat sink B has the highest temperature, 122ºC, and follow by heat sink B, 81ºC, and then heat sink A, 71ºC. In a nut shell, the lower is the volume area, the higher is the rate of heat transfer, the higher the final temperature.

It is worth to mention that the maximum temperature here does not occur at the heat sink, but at the GPU chip because the latter is the heat source. A heat sink has the highest temperature in the whole system. If the temperature in the contour is lower, more heat is removed from the GPU chip to the heat sink, it shows that the heat sink is more efficient.

STREAMLINE

Figure 10 shows the velocity streamline at speed 1 ms^{-1} and 2 ms^{-1} for the steady and transient state in CPU inlet and outlet. The streamline for inlet speed 1 ms^{-1} and 2 ms^{-1} is almost similar. The only difference is the velocity magnitude and the "vortex" at the above of the CPU case. For both speeds, the air enters the CPU case at high speed, after it passes through the heat sink to remove the heat and travels out the CPU case outlet, with high speed again. The "vortex" generation at higher speed is more compact when compared to low speed due to the pressure distribution in the CPU case is different.

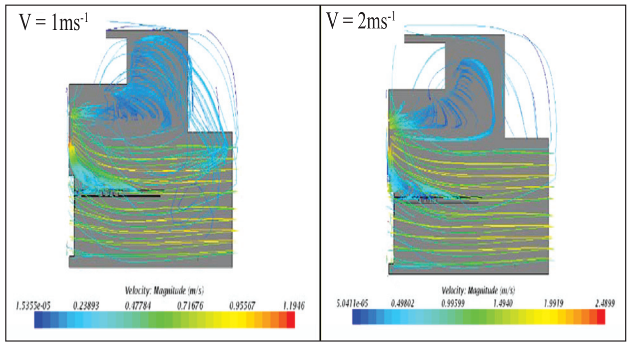


FIGURE 10. Streamline at speed 1 ms^{-1} and 2 ms^{-1}

TEMPERATURE DISTRIBUTION WITHIN TIME INTERVAL

Figure 11, shows the temperature distribution in the CPU case due to convection with the increase of time interval. In the beginning, the heat release by graphics card focuses on the graphic card surrounding the area. After some time, the heat slowly spread to the upper part of the CPU case via convection process. This phenomenon occurs because the density of the hot air is lower than the cool air. Hence, the hot air will keep focus at the upper part of the CPU case with the increment of the time interval.

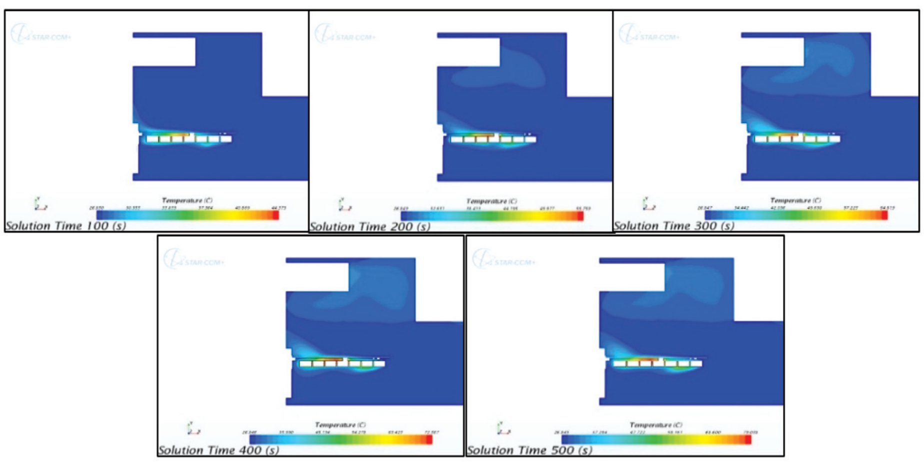


FIGURE 11. Temperature distribution in CPU case with time interval t = 100 s, 200 s, 300 s, 400 s, 500 s

SUGGESTED SOLUTIONS

In order to increase the cooling system for CPU problem, there are some methods here, such as increase the height and number of heat sink fin to increase the total surface area, Figure 12(a). Next, allocate some fan at the upper part of the CPU case to reduce the heat accumulate at the upper part of CPU case and at the same time the power supply of CPU which located at the upper part should change to lower part to increase the upper area for speeding up the rate the heat released, Figure 12(b). Increase the size of inlet region and the number fan and speed to increase the magnitude and velocity of wind inlet to speed up the convection process, Figure 11(c).

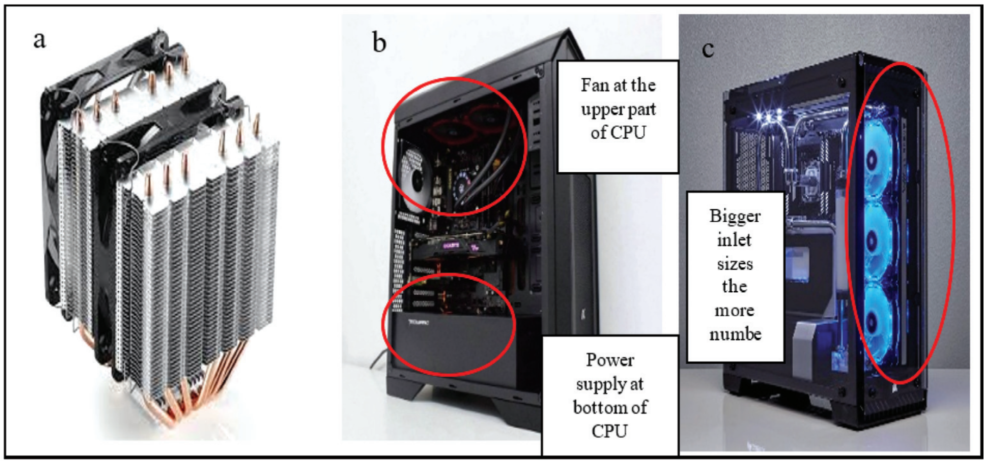


FIGURE 12. Suggestion solution for computer case cooling system

CONCLUSION

We conducted numerical simulations for heat transfer performance that occurs via conduction and convection modes. Conduction occurs in GPU chip to heat sink due to the GPU chip and the other component in contact with the heat sink. Convection occurs inside CPU case due to the heat transfer from the heat sink to the air inside CPU case via air inlet and outlet. Form the result, the simulation solves and show temperature distribution and heat transfer in the heat sink and inside computer case in both steady state and transient state successfully. These results can help in propose and solve the overheating temperature of GPU issue. The following conclusions are drawn from the current article:

1. In this paper, the authors presented the effect of changing the shape and area of the heat sink to increase the rate of heat transfer according to the speed in the simulation results.
2. The comparison of temperature distribution contour of each heat sinks for speed 1ms^{-1} at a different time interval (t = 100 s, 200 s, 300 s, 400 s, 500 s). Based on the contour temperature distribution, shown that heat sink B has the fastest rate of heat exchange followed by heat sink C and heat sink A
3. The "vortex" generation at a higher velocity is more compact when compared to low velocity due to the pressure distribution in the CPU case is different when the velocity streamlines at a different speed such as 1 ms^{-1} and 2 ms^{-1} for the steady and transient state.
4. Temperature distribution in CPU case with different time interval such as t = 100 s, 200 s, 300 s, 400 s, 500 s, it shows that due to convection with time, In the starting the heat release by graphics card focuses at the graphics card surrounding the area, afterwards the heat slowly spread to the upper part of the CPU case via convection process.

ACKNOWLEDGEMENTS

We would like to express our gratitude for the financial supports provided by the Ministry of Higher Education (MOHE) through fundamental research grant FRGS/1/2016/TK03/UKM/02/1 and UKM grant GUP-2018-102.

REFERENCES

Abdullah, M.F., Zulkifli, R., Harun, Z., Abdullah, A. & Ghopa, W.W.A. 2017. Studying of convective heat transfer over an aluminum flat plate based on twin jets impingement mechanism for different reynolds number. *International Journal of Mechanical and Mechatronics Engineering* 17(6): 1-16.

Bejan. 2013. *Convection Heat Transfer* .Ed. John Wiley & Sons.

Bolgarskii, A., Mukhachev, G. & Shchukin, V.K. 1967. Thermodynamics and Heat Transfer. Defence Technology Information Center

Chai, L., Guodong, X., Liang, W., Mingzheng, Z. & Zhenzhen, C. 2013. Heat transfer enhancement in microchannel heat sinks with periodic expansion-constriction cross-sections. *International Journal of Heat and Mass Transfer* 62(1): 741-51.

Ferrari, A. & Suppa, V. 1977. Cooling device for components which dissipate large amounts of heat, Google Patents.

Incropera, F.P., Lavine, A.S., Bergman T.L. & DeWitt, D.P. 2007. *Fundamentals of Heat and Mass Transfer.* Wiley.

Khabari, A., Zenouzi, M., O'Connor, T. & Rodas. A. 2014. Natural and forced convective heat transfer analysis of nanostructured surface. *Proceedings of the World Congress on Engineering.* 1.

Leach, T.T. & Cadou, C.P. 2005. The role of structural heat exchange and heat loss in the design of efficient silicon micro-combustors. *Proceedings of the Combustion Institute* 30(2): 2437-44.

Lee, S.C. 2003. *Heat Sink Device for Cooling Chipset*. Google Patents.

Li, W., Minghe, X., Jing, R. & Hongde, J. 2017. Experimental investigation of local and average heat transfer coefficients under an inline impinging jet array, including jets with low impingement distance and inclined angle. *Journal of Heat Transfer-Transactions of the ASME* 139(1): 12201.

Liu, Y., Cui, J., Jiang, Y.X. & Li. W.Z. 2011. A Numerical study on heat transfer performance of microchannels with different surface microstructures. *Applied Thermal Engineering* 31(5): 921-31.

Abdullah, M.F., Harun, Z., Abdullah, S., Ghopa, W.W.A. & Zulkifli, R.. 2017. Experimental investigation on comparison of local Nusselt Number using twin jet impingement mechanism. *International Journal of Mechanical & Mechatronics Engineering IJMME-IJENS* 17(04): 60-75.

Martin, J. 1995. *Combination Conductive and Convective Heat Sink*. Google Patents.

Kuppusamy, R., Navin, R.S., Ghazali, N.N.N. & Mohammed, H.A. 2014. Numerical study of thermal enhancement in micro channel heat sink with secondary flow. *Interational Journal of Heat and Mass Transfer* 78: 216-23.

Shah, R.K. & Sekulic, D.P. 2003. *Fundamentals of Heat Exchanger Design*. John Wiley & Sons.

Smith, J.M. 1975. Introduction to Chemical Engineering Thermodynamics. Tesis Rensselaer Polytechnic Institute.

Bergman, T.L., Lavine, A.S., Incropera, F.P. & Dewitt, D.P. 2011. *Fundamentals of Heat and Mass Transfer*. United States of America.

Xia, G., Chai, L. Zhou, M. & Wang. H. 2011. Effects of structural parameters on fluid Flow and heat transfer in a microchannel with aligned fan-shaped reentrant cavities. *International Journal of Thermal Sciences* 50(3): 411-19.

Xia, G., Zhai, Y. & Cui. Z. 2013. Numerical Investigation of thermal enhancement in a micro heat sink with fan-shaped reentrant cavities and internal ribs. *Applied Thermal Engineering* 58(1-2): 52-60.

Yu, Y., Simon, T.W., Zhang, M., Yeom, T., North, M.T. & Cui, T. 2014. Enhancing heat transfer in air-cooled heat sinks using Piezoelectrically-Driven sgitators and synthetic jets. *International Journal of Heat and Mass Transfer* 68(0): 184-93.

Yunus, A. C. & Michael, A. B. 2006. *Thermodynamics: An Engineering Approac*h. New York: McGraw-Hill.

*Zambri Harun, Ng Jie Suang, W. M. Faizal W. Mahmood, Mahir Faris Abdullah, Eslam Reda Lotfy,
Centre for Integrated Design for Most Advanced Mechanical System
Universiti Kebangsaan Malaysia,
43600 UKM Bangi, Malaysia