VANOS
JOURNAL OF MECHANICAL ENGINEERING EDUCATION
http://jurnal.untirta.ac.id/index.php/vanos
ISSN 2528-2611, e-ISSN 2528-2700
Vol.2, No.2, December 2017, Page.97-104.

NUMERICAL COOLING SIMULATION ON LAPTOP HEAT SINKS WITH VARIATIONS OF DIFFERENT AIRFLOW SPEEDS

Syamsuri[1], Novi Indah Riani[1]
[1]Department of Mechanical Engineering, Adhi Tama Institute of Technology Surabaya
Arief Rahman Hakim St., No. 100, Surabaya, 60117
syam_sby2003@yahoo.com

Accepted: 7 November 2017. Approved: 30 November 2017. Published: 30 December 2017

ABSTRACT

Cooling on electronic objects, especially laptop is something to note. If the cooling system is not good, there will be overheating on the main chip motherboard and other components so that the laptop will be damaged quickly. This research aims to determine the temperature distribution on the heat sink laptop so that overheat can be overcome. This research was carried out numerically, where the specimen was a heat sink component on a laptop unit simulated using Solid works 2011 software in three dimensions with variations carried out in the form of air velocity values passing through the specimens of 5, 10, and 15 m/s. The results obtained in this study showed that the higher the speed of air flow through the heat sink, the higher the transfer of heat transfer.

Keywords: Heat sink, solid works 2011, airflow velocity variation, heat transfer.

INTRODUCTION

Improved capabilities in today's modern computer technology cause the electronics components to become hotter faster. This problem can degrade the performance of the component. Along with the problem, then the need for an effective cooling tool to reduce the heat on the component becomes higher. Cooling using a heat sink is one solution that is quite interesting.

Heat sinks are cooling devices commonly used today. Given the importance of the role of heat sink, its performance needs to be analyzed. Research on heat sink has been done by many researchers before. Sajedi, et al (2016) performed numerical analysis to determine the effect of splitters on the hydro-thermal behavior of pin fins. The results showed that splitter use improved the hydro-thermal performance of both pins.

Flat heat sink with a circular shape decreases splitter pressure by 13.4%, and thermal resistance decreases by 36.8%. While the square shape of the splitter pressure decreases of 8.5%. The researchers then examined the efficiency of fluid cooling on heat sinks that contained a pin microfine. With an analysis of the geometry and pressure drop on the pin diameter performed, it was found that effective thermal resistance would achieve an optimum value of 0.26 ˚C / W occurring between the pivot range of 0.5 and 0.6 (Chiu, et al, 2017). Based on research that has been done by the researchers before, came the thought to do the efficiency of cooling on the heat sink. One of them is done by looking at the value of the temperature distribution that occurs in the heat sink area to the speed of the airflow through it.

RESEARCH METHODS

The research is done by drawing the design of the workpiece that will be made on the solidworks software part 2011. From the image file will be transferred to the solidworks 2011 software assembly to assemble every component of work piece that has been part or drawn. From the assembly or assembly of each workpiece component will get the results in accordance with the design of the drawings that have been made. The geometry scheme can be seen in the picture below:

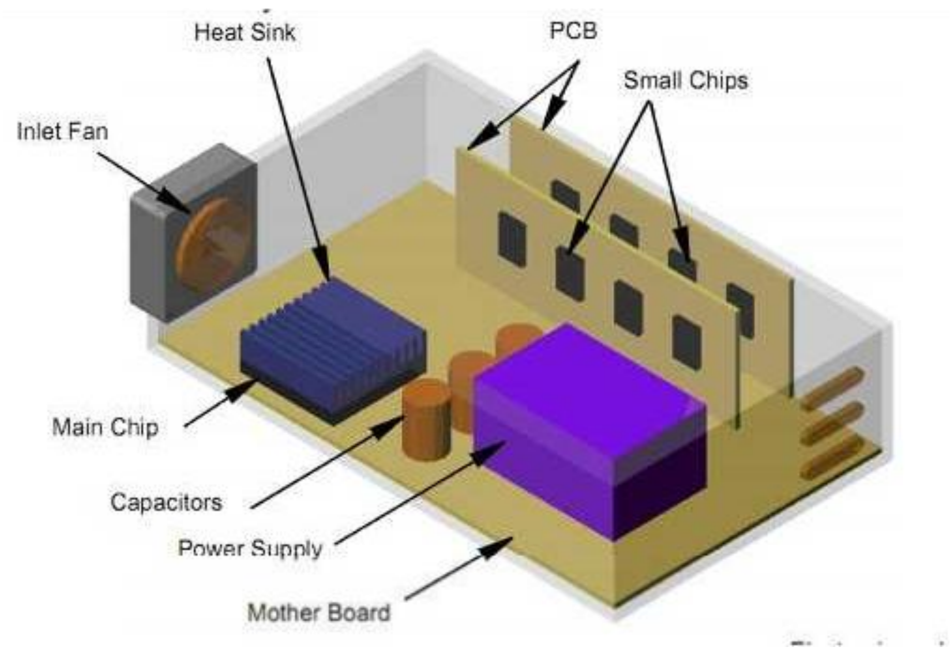


Fig 1. Preparation of flow simulation project model

Stages performed in this study are divided into three stages, namely:

A. *Pre-processing*

The pre-processing stage is the stage where the data is inputted from geometry dimension, model taking into small elements (meshing), defining domains and boundary conditions in Solid works 2011. The geometry model consists of part, assembly, mate, assembly, and check geometry.

B. *Processing*

At this stage include all the necessary data and ensure that the geometry to be simulated is correct. The flow rate is varied in the flow simulation model of 5, 10, and 15 m / s. Pressure inlet and pressure outlet using ambient pressure.

Define heat source is 5W, and finally, prepare the project calculation.

C. *Post-processing*

Post processing is the process of displaying results and analysis of the results obtained. The data to be taken in the form of temperature distribution with a variety of fluid flow velocity given to the heat sink.

RESULTS AND DISCUSSION

The focus of this research is to know the speed of air that can reduce heat at the most optimal heat sink. The data displayed in the form of contours of temperature visible above, front and side, and temperature distribution on the heat sink.

Numerical Validation

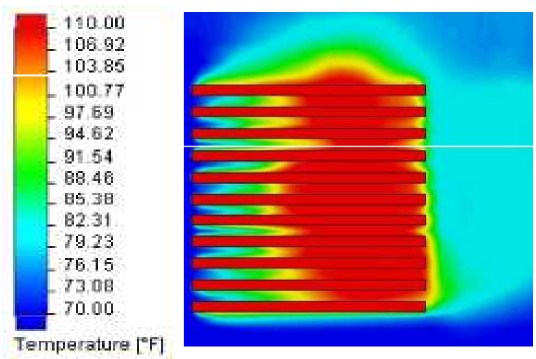
Before doing post processing, numerical validation of meshing is required. Data of numerical simulation results compared with data performed by (Anuyasha, et al, 2014).

Table 1. Maximum Temperature Value

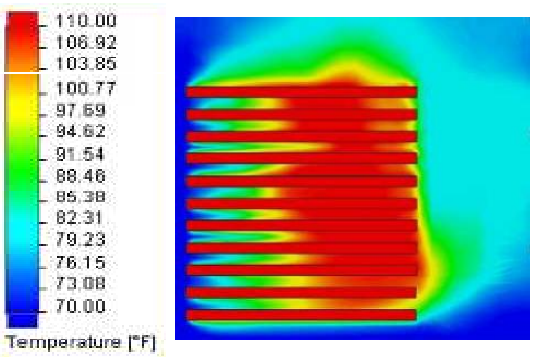
Object		Anusaya.M et al, 2014	Andre Hnyeur 2015	There is a difference %
Main Chip	Tmax °F	114.1 °F	113.8 °F	**2.6 %**
Heat Sink	**T max °F**	**114.1 °F**	**113.8 °F**	**2.6 %**

From table 1 above, it can be seen that the results of this simulation compared with the previous research showed that there is a suitability, where the difference is 2.6% or less than 10%. This shows that this simulation is capable of capturing the physical phenomenon that happened.

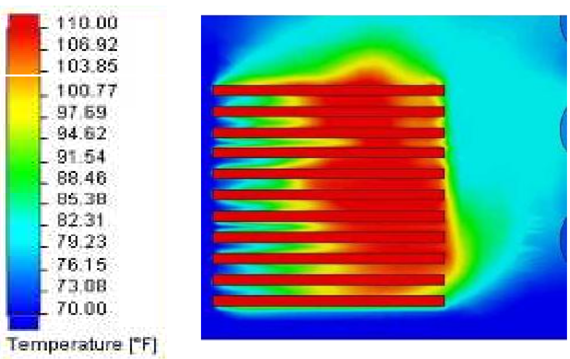
A. Contour Temperature of Heat Sink Top View



(a)



(b)

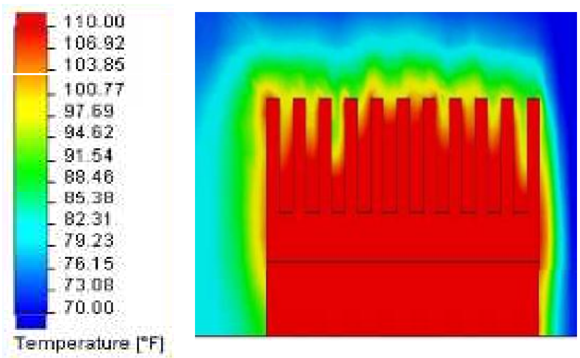


(c)

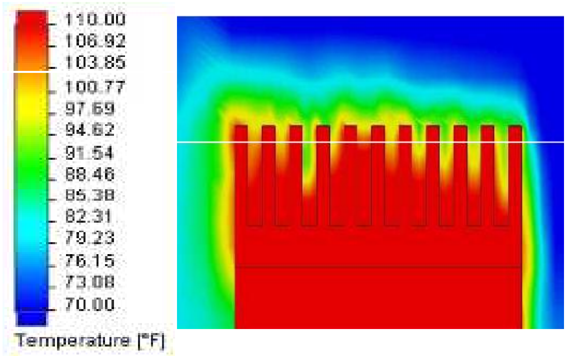
Fig 2. The contour of the heatsink temperature appears over with the speed (a) 5 m / s, (b) 10 m / s, and (c) 15 m / s.

In Figure 2 above shows the temperature contours of the upper heat sink, its position on the Z axis. Figure 2 (a) shows the heat transfer time occurring on the heat sink surface is 00:05 sec. While 2 (b) and 2 (c) are each at 00:07 and 00:09 seconds. In the three pictures, it is seen that the temperature in the heat sink is greater than the temperature of the fluid around the area of heating that occurs behind the heat sink. This is because cooling dominates the area in front of the heat sink because the cold fluid enters from front to back. This phenomenon is called the convection phenomenon of the heating temperature of the heat sink surface to the cold fluid flowing through the heat sink. The cooling process on the three contours of the above temperatures appears significant, but it can be seen that at the given 15 m / s air velocity shows that cooling is better.

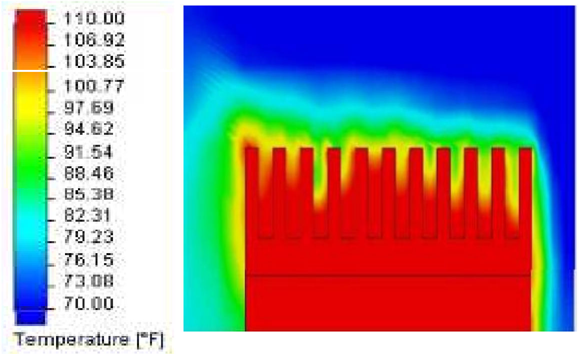
B. Contour Temperature of Front View Heatsink



(a)



(b)

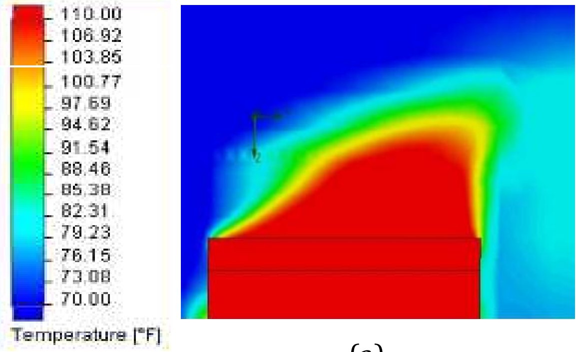


(c)

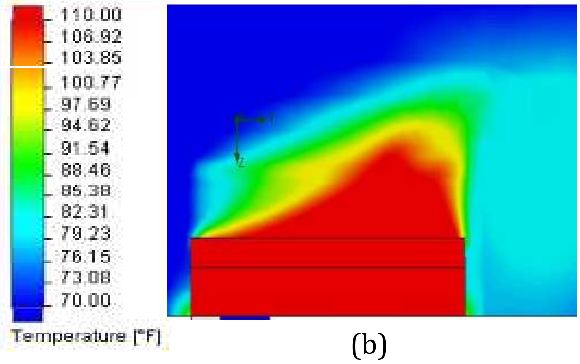
Fig 3. Contour temperature of the heat sink front view with speed (a) 5 m/s, (b) 10 m/s, and (c) 15 m/s

The picture of the temperature contour of the front of the heat sink is shown in Figure 3. The position of the front heat sink position is on the Y-axis, and the heat transfer time occurring on the front heat sink is 00:01 seconds, 00:03 seconds, and 00:05 seconds for each (a), (b), and (c). In the three pictures, it is seen that the temperature on the surface of the uniform heat sink and dominates the entire surface. Maximum temperatures are present on the surface of the heat sink marked in red, while the minimum temperature is denoted by blue to the surrounding fluid. Convection heat transfer occurs between the surface of the heat sink with a fluid that has a low temperature around it.

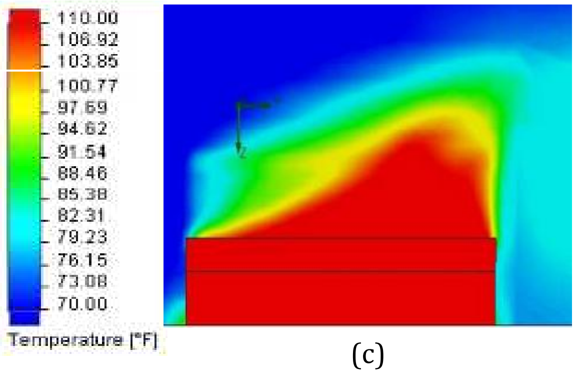
C. Contour Temperature of Heat Sink Side View



(a)



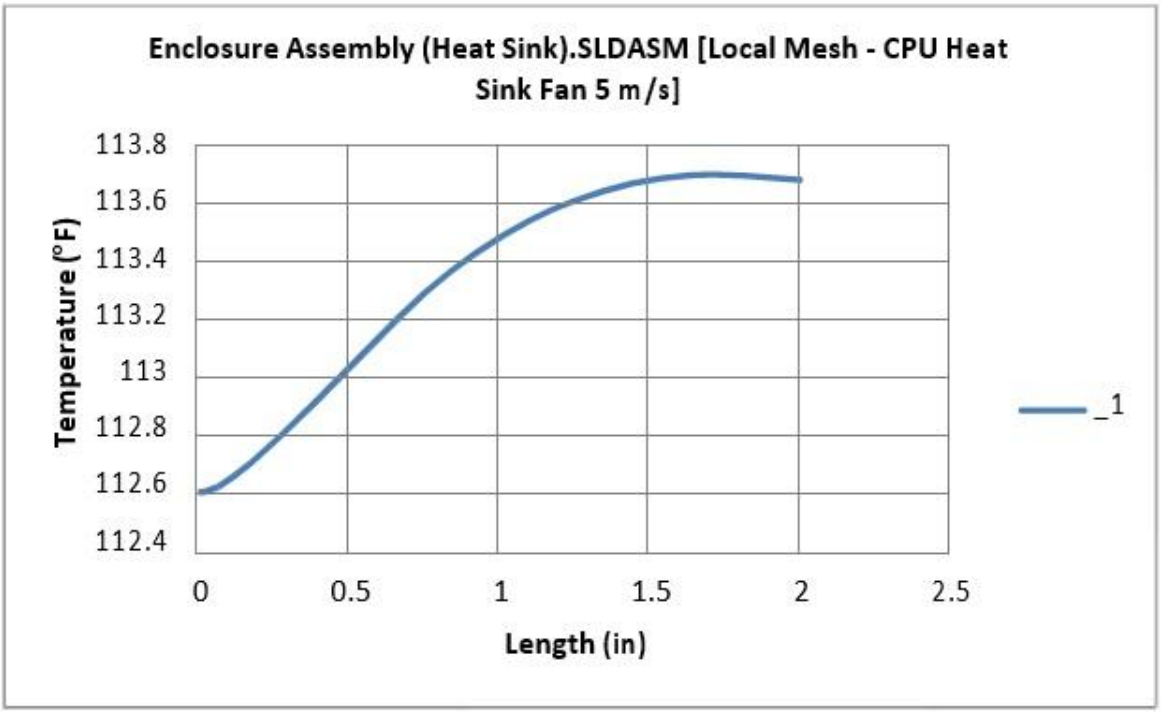
(b)



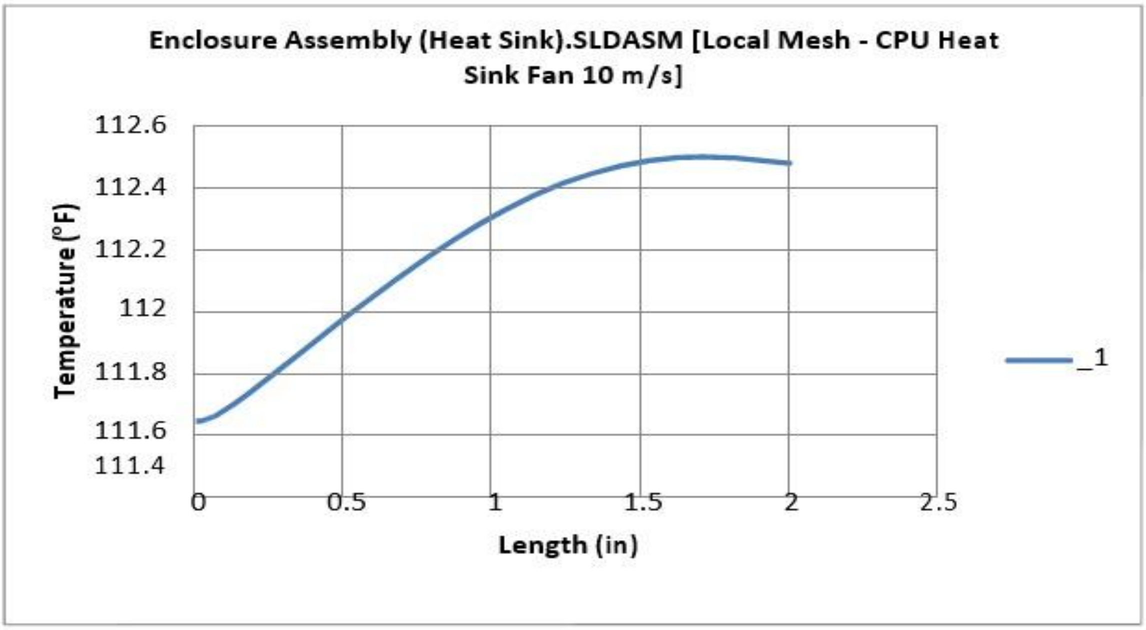
(c)

Fig 4. Contour side heatsink temperature contour with speed (a) 5 m/s, (b) 10 m/s, and (c) 15 m/s

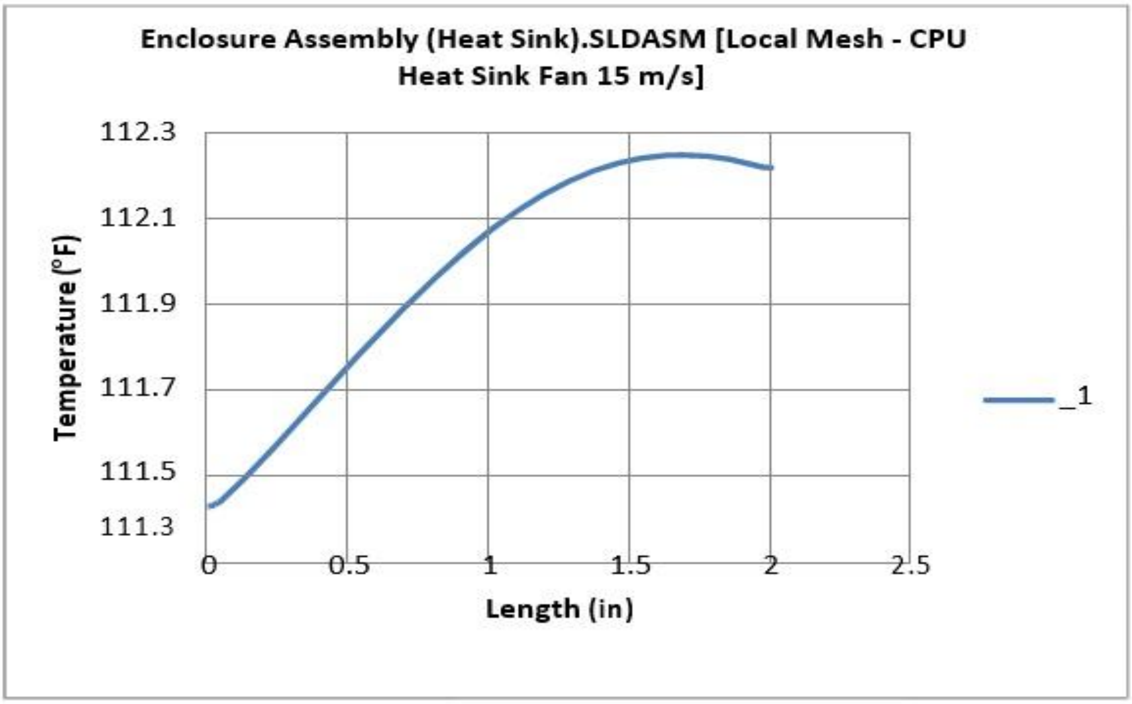
D. Temperature distribution in heat sink at center location



(a)



(b)



(c)

Fig 5. Distribution in the heatsink at the center location for air input speed: (a) 5 m/s, (b) 10 m/s, and (c) 15 m/s.

In that picture shows that the temperature distribution is increased with increasing distance. This is due to the convection coefficient is influenced by flow velocity (Anuyasha, et al, 2014). In the initial data retrieval position that is X = 0 in the area, there is a narrowing of the heat transfer area. According to the theory of continuity (Philip and McDonalds, 2011) with the same discharge but the area is smaller, the velocity will enlarge (Q = VA), with a velocity equal to X = 0, the coefficient of heat transfer will decrease and cause the temperature in area X is also small. The more the distance increases or the distance increases from X = 0 to X = 2, then the temperature increases.

Table 2. The relationship between Air Flow Variation in Heat Sink and Maximum Temperature.

Object	Unit	*Maximum value*
Heat sink speed 5 m/s	[°F]	**133.8**
Heat sink speed 10 m/s	[°F]	**112.7**
Heat sink speed 15 m/s	**[°F]**	**111.6**

In that table, it is seen that with the higher air velocity given, then the maximum temperature is lower. This is because the speed affects the coefficient of convection heat transfer. This study is consistent with research conducted by (Anuyasha, et al, 2014). The higher the airflow rate the higher the cooling occurs (Incropera and Dewitt, 1996).

CONCLUSION

Based on the analysis that has been done about the effect of air velocity variation on cooling on the heat sink, it can be concluded:

1. From the temperature contour, it is seen that the heating temperature accumulates at the rear of the heat sink for the upper case, and the heat transfer time occurring on the heat sink surface is 00:05, 00:07, 00:09 s for 5, 10, 15 m/s.
2. For the case of a heatsink side view of hot temperature contours following the boundary layer theory of temperature, the heat transfer time is 00:02, 00:03, 00:04 seconds for different speeds.
3. While the temperature contour of the front heatsink looks that the heating temperature covers the entire area of the heat sink, the heat transfer time that occurs at each speed is 00:01, 00:03, 00:05 seconds

Thus, with variations in the air velocity provided by the heat sink can speed up the cooling process. This is because the heat transfer process, especially by convection, in addition to being influenced by the surface area, is also affected by the forced convection. In this study, forced convection is provided in the form of air velocity passing through the heat sink.

REFERENCES

Anusaya M,Salwe, Shubham,S pande Jouned, S khan., (2014) '*Effect of velocity and particle size on the coefficiendt of Heat transfer in fluidized Bed Heat Excharge'*, International Jurnal of Enggineering Research and general Science,Volume 2, Issue2 , Feb-Maret 2014.

Chiu, Han-Cieh., Hsieh, Ren-Horn., dkk, (2017), *'The Heat Transfer Characteristics of Liquid Cooling Heat Sink with Micro Pin Fins'*, Journal of International Communication in Heat and Mass Transfer 86 (2017) 174-180.

Incopera.F.P.and Dewitt D.P, (1996), Fundamentls of Heat and Mas Transfer, John wiley and Sons.

Philip J. Prichard Fox and McDonalds, (2011), Introduction to Fluid Mechanics 8th Edition, America.

Sajedi. R., kk., (2016) *'Spiltter Plate Application onThe Circular and Square Pin Fin Heat Sinks'*, Journal of Microelectronics Reliability, MR-11972 No of Pages 11.