IJARI

International Journal of Advance Research and Innovation
Vol. 8(4), Oct-Dec 2020, pp. 159-167
Doi: 10.51976/ijari.842026

Article Info

Received: 29 Jul 2020 | Revised Submission: 20 Oct 2020 | Accepted: 28 Oct 2020 | Available Online: 15 Dec 2020

Study of Active Plate Heat Sink using Different Slope Angles

Amanbir Sokhi, Harpuneet Singh** and Harpreet Singh***

ABSTRACT

In recent years, cost-effective compact size devices with high processing speed are made for a sharp rise in the electronics market. To compensate for the high heat fluxes produced in the device, heat sinks are extensively used. In this research, study of active plate heat sink using different slope angle without changing surface area and mass is carried out with the help of CFD tool. The methodology used in this study is the 3D CAD modeling using SOLIDWORKS software and CFD simulation using STAR-CCM+ software. Navier-Stocks equations are solved by using the conjugate heat transfer method. To establish a validation case, the numerical result obtained from this study is compared with the experimental results of the previous research paper. From the results, maximum temperature and thermal resistance performance of 7.5° slope geometry is increased by 1.57% and 16.87% respectively and pressure drop performance of 10° slope geometry is increased by 65.37%. Another considerable factor is the temperature profile at the bottom of the sloped heat sink geometries, which is more uniformly centered in case of 7.5° and 10° slope. The conclusion stated that the angular sloped geometries provides better result than the traditional plate heat sink geometry.

Keywords: *Computational fluid dynamics; Active heat sink; Performance parameters; Slope angle; Governing equations.*

1.0 Introduction

In the modern era of technological advancement, high-speed processing electronic devices plays an important role in our day-to-day life. To fulfill the purpose to cool down these electronic devices, heat sinks are extensively used. In recent years, due to sharp rise in the electronics market, cost effective compact size devices with high processing speed is the need of the hour. Due to compactness of the devices, there is a sharp rise in power density which produces high heat fluxes in the device [1]. Removal of extensive heat from the device and ensures uniform power dissipation inside the device puts tremendous pressure on the engineers to make a cost effective highly compact heat sink design. Research shows that for every 10 °C rise in operating temperature over the junction temperature, electronic devices failed at the doubled rate [2]. So to efficiently cool the electronic device, the geometry design of the heat sink is an important parameter.

Heat Sink generally is a heat transfer thermal conductive metal component mainly used to absorb and dissipate heat away from the high temperature electronic devices. It works as a cooling device and protects the electronic devices from being damaged due to high temperature. The main component of electronic devices is the processor chip, which processes calculation at a very high speed, due to which the temperature of the chip rises at a very rapid rate. To control the temperature of the processor chip, heat sink is attached to the processor chip to maintain its temperature and protects it from damaging. The heat sink is directly attached to the processor chip to perform its function.

Heat Sink mainly consists of fins which are attached to the base of heat sink. Fins are the thin pieces of metal attached to the base of heat sink which helps to dissipate heat from the electronic device through a much larger surface area and the process is called conduction. Air generally surrounds the heat sink fins to help reduce the temperature of heat sink through natural convection. In most of the

**Corresponding author; Department of Mechanical Engineering, IK Gujral Punjab Technical University, Jalandhar, Punjab, India (E-mail: amanbirsingh1992@gmail.com)*

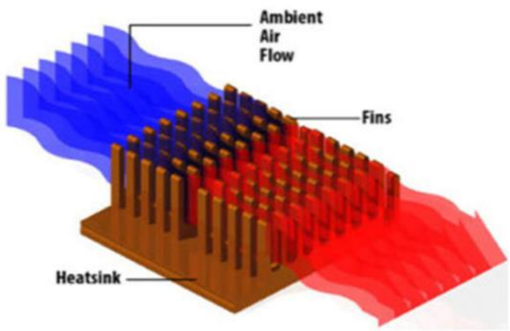
***Department of Mechanical Engineering, Khalsa College of Engineering & Technology, Amritsar, Punjab, India*

high purpose processor chip, fan is also used to increase the phenomenon of convection and the process is called forced convection.

Due to the development in the computer software industry, computational fluid dynamics (CFD) is the modern approach among researchers, engineers and designers to draw results on problems without wasting money and time on experiments. CFD numerical simulation tool is extensively used in fluid flow behavior, heat and mass transfer related problems like heat exchanger, heat sink etc., phase change, chemical reaction, mechanical movement and stress strain in structures like problems

[3]. There are many CFD software packages are available in market like ANSYS-FLUENT, STARCCM+ and many more. Lots of researcher used these simulation software to draw numerical results by validating data with the experimental data and conduct further improvement study, using same conditions without experiments. CFD tools when used in case of heat sink problem, provides good and well oriented results [4]. In this study, CFD simulation on active plate-fin heat sink is carried out to check the performance parameters using different slope angles with the traditional plate heat sink.

Figure 1: A Typical Heat Sink



2.0 Literature Survey

Arularasan and Velraj, 2008 choose an optimal heat sink design, CFD modeling and simulations were conducted in preliminary studies of the fluid flow and the heat transfer characteristics of the parallel plate heat sink by using the commercial package provided by Fluent Inc [5]. This work has been analyzed by the geometric parameters fin height, fin thickness, base height and fin pitch. The ideal configuration of the heat sink on a parallel plate heat sink using a CFD analysis is carried out in this research project. Experimental studies with a parallel heat sink plate have been performed to verify the heat sink model. The design engineers who are interested in the cooling of electronics by using the plate heat sink benefit from these tests and conclusions. Sivasankaran et al., 2010 have studied the parallel plate fin and cross cut pin fin heat sinks where the heating element is asymmetrically positioned [6]. For various heat sinks, the thermal performance, coefficient of heat transfer and efficiencies were compared. It is found that the average coefficient of heat transfer of the parallel plate fins is higher than that of the cross cut pin fins. In the experimental research, the fan distance is varied to find the optimum distance for maximum efficiency for both parallel plate and heat sinks with cross cut pin fin.

Rodgers and Eveloy, 2013 presented the overview of analytical, semi-empirical and Computational Fluid Dynamics (CFD)-based methodologies for air-cooled heat sink design optimization in free convection [7].They discussed the benefits and possible shortcomings of such design techniques in their research. A multifaceted design approach that blends these modeling approaches with experimentation has been advocated. The study suggested that the progress in heat sink architecture should put emphasis on sustainability and formal optimization in the near future.

Gupta et al., 2014 focused on CFD and thermal analysis of rectangular plate fin and cylindrical pin fin heat sinks with a key focus on temperature distribution and heat flux [8]. The results of this work showed that the total heat transfer rate of rectangular plate fins is greater than predicted for cylindrical pin fins with the same dimensions and boundary conditions. Haghighi et al., 2018 performed an experimental analysis to calculate the coefficient of convective heat transfer and thermal performance of plate and plate cubic pin-fin heat sinks with a natural convection process [9]. The analysis was carried out for Rayleigh number from 8000000 to 9500000 and 10W to 120 W input power. Fin spacing and fin numbers differ between 5–12 mm and 5–9 mm, respectively. The results showed that plate cubic pin heat sinks have lower thermal resistance and higher heat transfer relative to plate2heat sinks. Improvement of heat transfer of newly built heat sinks is 10–41.6% higher than normal pin-fins. Increased fin spaces in all forms of heat sinks studied cause lower thermal resistance. Increasing fin numbers, however, does not result in better heat transfer. A plate cubic pin-fin heat sink with 7 fins and 8.5 mm fin spacing was the best design for the heat sink.

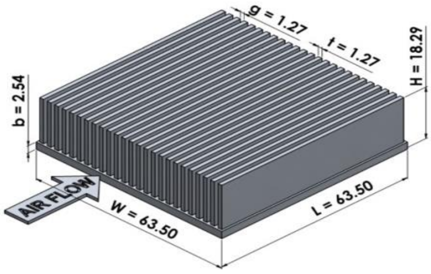
3.0 Methodology

The methodology used for computing the results of different plate heat sink geometries and the validation case depends upon the two main pillars which are 3D Cad Modeling and CFD (computational fluid dynamics) analysis. CAD Modeling was done with the help of SOLIDWORKS software due to its user friendly nature and cost effective advantages over other software. Numerical simulation was done by using CFD tool STARCCM+ software for the analysis purposes due to its multi-physics real world problem solving nature with accuracy and easy automatic meshing generally in case of heat sinks.

Table 1: Heat Sink Geometry Specification for Simulation

Design Notation	Dimensions	Condition
Width (W)	63.5mm	Constant
Length (L)	63.5mm	Constant
Height (H)	18.29mm	Vary (according to the design)
Fin Thickness (t)	1.27mm	Constant
Channel Width (g)	1.27mm	Constant
Base Thickness (b)	2.54mm	Constant
Number of fins (n)	25	Constant

Figure 2: 3D CAD Model of Plate Heat Sink with Dimensions



3.1 CAD modeling

The dimensions used in this study for making the geometries of the plate heat sink are different from the validation case geometry dimensions to simplify the study results. Following table shows the dimensions used in the 3D cad software for making the geometries of heat sink.

While making others geometries of the heat sink, keep in mind the mass and surface area remains constant so that the performance parameters depends only on the shape of geometries of the heat sink and not on the size. To do this, following conditions must satisfy while making the design of the other geometries of the plate heat sink-

- The dimensions of the base of the heat sink remain constant for all geometries.
- While changing shape, the height of the fins vary in such a way that the overall mass of the geometry remains constant.

3.2 CFD simulation

The CFD simulation consists of three stages:

3.2.1. Pre-processing

Pre-processing is the first stage of CFD simulation process which is performed before doing the actual simulation. It is like preparing the food before eating. It consists of the CAD geometry preparation, defining the solution domain and regions, selecting and preparing the meshing and physics model and define the initial and boundary conditions.

3.2.2 Conjugate heat transfer (CHT) method (Definition of regions)

Due to the presence of both solid region in the form of heat sink and fluid region in the form of air force to use the conjugate heat transfer method for the simulation run. The analysis type Conjugate heat transfer (CHT) allows for the simulation of heat transfer between Solid and Fluid domains by exchanging thermal energy on interfaces between solid and fluid domains.

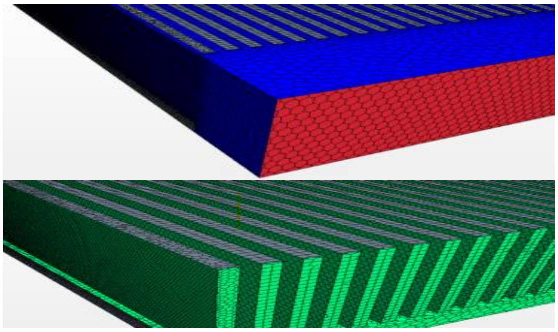
3.2.3 Meshing

Table 2: Meshing models used in this study.

Embedded thin mesher
Extruder
Generalized cylinder
Polyhedral mesher
Surface remesher

Using STARCCM+ software there are different meshers which are used to divide the geometry of the two regions depending upon the size and shape of the geometries are given in the table 2 above. The base size is set at the value of 1.5mm for all heat sink geometries and 5mm for the geometries of validation case. In this study, the geometries used were not too complex for meshing so there is no need to alter any other condition. Also with increase in the base size, the accuracy of the study increases but cost and time of the study also increases.

Figure 3: Meshing of Plate Heat Sink with and Without Fluid Region



3.3 Physics models used for fluid and solid

Physics models are the assumptions made to relate the CFD problem with the real world problem so that the accuracy is maintained in the simulation process.

Table 3: Physics Models for Fluid

Fluid	Comments
Three dimensional	Movement of fluids in all three directions
Steady state	Not dependent on time
Gas	Generally as Air
Segregated Flow	The segregated flow model invokes the segregated solver which solves each of the momentum equations in turn, one for each dimension.
Gradients	Gradients are used in several places within the finite volume solution methodology.
Laminar	Reynolds number less than 2200
Ideal Gas	The collision between molecules assumes to be elastic.
Segregated fluid temperature	The Segregated Fluid Temperature model solves the total energy equation with temperature as the solved variable.
Gravity	In the negative Y direction

Table 4: Physics Models for Solid

Solid	Comments
Three dimensional	Movement of fluids in all three directions
Steady state	Not dependent on time
Solid	Generally as Aluminium
Segregated solid energy	Include both secondary gradients.
Gradients	Gradients are used in several places within the finite volume solution methodology.
Constant Density	Density does not change throughout.

3.4 Reynolds number

Reynolds number is dimensionless number used to predict the flow nature of the fluid while flowing through the channel of the pipe or object. The flow may be laminar, transition and turbulent depending upon the value of Reynolds number. In case of laminar flow the Reynolds number should be less than 2200 and in case of turbulent flow the Reynolds number should exceed 4000. In between these values the flow may be in transition stage and possesses properties of both turbulent and laminar flow. The formula used to define the Reynolds number is given as

$$Re = \frac{\rho v D}{\mu} \qquad \ldots(1)$$

To find out the Reynolds number used in the heat sink simulation, putting all the values in the formula. To find out the Reynolds number used in the heat sink simulation, putting all the values in the formula

$$D = \frac{4A}{P} = 0.025m \qquad \ldots(2)$$

The value of Area (A) and perimeter (P) is calculated from the heat sink geometry, so the value of Reynolds number (Re) is

$$Re = \frac{\rho v D}{\mu} = \frac{1 \times 1 \times 0.025}{1.85508 \times 10^{-5}} = 1348 \qquad \ldots(3)$$

The value of density of fluid (ρ), velocity of fluid (v) and Dynamic viscosity (μ) remains same throughout the study so the flow remains laminar in nature for all the geometries.

3.5 Initial and Boundary conditions

Table 5: Boundary Conditions for Fluid Region

Boundary Face	Boundary Condition	Comment
Inlet	Velocity Inlet	Value = 1m/s
Outlet	Pressure Outlet	Split Ratio value = 1.0
Left	Symmetry Plane	-
Right	Symmetry Plane	-
Top	Symmetry Plane	-
Bottom	Wall	Adiabatic
Default	Wall	Adiabatic contact wall between fluid and solid.

Table 6: Boundary Conditions for Solid Region

Boundary Face	Boundary Condition	Comment
Solid Bottom	Wall	Heat source value = 100Watt
Default	Wall	Adiabatic contact wall between fluid and solid.

3.6 Solver

During simulation process, solvers in Simcenter STAR-CCM+ calculate the solution.

3.6.1 Governing equations

The governing equations of fluid dynamics are the conservation laws of mass, momentum, and energy. In case of CFD, these set of conservation laws are called Navier–Stokes equations. These equations are derived from the Newton's second law of motion under fluid in motion and stresses in the form of partial differential equations. In this study, these equations are used to simulate the fluid behavior around the heat sink. These equations are very complex in nature, so to solve these equations high level of computational power is required. The assumptions made to solve these equations are stated below:

a) conjugate heat transfer method that is solid fluid combination is used.
b) the air flow around the heat sink is three-dimensional, steady state, incompressible and single phase in nature.
c) the flow is laminar in nature.
d) the densities of air and aluminium remain constant throughout.

According to these assumptions the equation of mass, momentum and energy are given as-

The equation of continuity is given by,

$$\nabla.(\rho\vec{U}) = 0 \qquad \ldots(4)$$

where as 'ρ' is the density of the fluid.

The momentum equation in three-dimensions is given by,

$$\nabla.(\rho\vec{U}\,u) = -\frac{\partial p}{\partial x} + \frac{\partial \tau_{xx}}{\partial x} + \frac{\partial \tau_{yx}}{\partial y} + \frac{\partial \tau_{zx}}{\partial z} \qquad \ldots(5)$$

$$\nabla.(\rho\vec{U}\,v) = -\frac{\partial p}{\partial y} + \frac{\partial \tau_{xy}}{\partial x} + \frac{\partial \tau_{yy}}{\partial y} + \frac{\partial \tau_{zy}}{\partial z} \qquad \ldots(6)$$

$$\nabla.(\rho\vec{U}\,w) = -\frac{\partial p}{\partial z} + \frac{\partial \tau_{xz}}{\partial x} + \frac{\partial \tau_{yz}}{\partial y} + \frac{\partial \tau_{zz}}{\partial z} \qquad \ldots(7)$$

where as '$\vec{U}$' is the fluid velocity with the components of u, v and w in x, y and z direction respectively, 'p' is the pressure and 'τ' is the tensor of viscous stress.

The energy equation is given by,

$$\nabla.(\rho h\vec{U}) = -p\nabla.\vec{U} + \nabla.(k\nabla T) + \Phi + s_h \qquad \ldots.(8)$$

where as h, k, Φ, T, and s_h are aggregate enthalpy, thermal conductivity, temperature, dissipation term and source term respectively.

These equations are solved on the basis of volume finite method in such a way that temperature and pressure are described by these equations.

3.7 Post-processing

The last step of the CFD simulation is to analyze the results by using different methods such as contour plot, vector plot, streamlines, and making reports using different variables.

a) Maximum temperature:

Maximum temperature is the maximum values of temperature at each boundary face of the solid and fluid region. The processor chip touches the bottom face of the heat sink so the maximum temperature is at the solid bottom of heat sink which was trying to keep low in this study. This temperature profile is used to find the thermal resistance of the heat sink.

b) Pressure drop profile

The average values of pressure of fluid at inlet and outlet was used to define the pressure drop around the heat sink. To calculate these values the report is made by created the derived planes at the inlet and outlet of the heat sink. The pressure drop is the difference in the inlet and outlet pressure which is given as,

$$\Delta P = P_{in} - P_{out} \qquad \ldots(9)$$

where as 'ΔP' is the pressure difference, 'P_{in}' and 'P_{out}' are the average pressure at the derived plane of inlet and outlet of the heat sink.

c) Thermal resistance

To find out the thermal resistance, the maximum temperature at the base 'T_{max}', fluid inlet temperature 'T_{inlet}' which is constant at 300K and the thermal specification of the heat source which is also kept constant at 100W is required. The maximum temperature of the heat sink varies according to the geometries of the heat sink. The thermal resistance is calculated by,

$$R = \frac{T_{max} - T_{inlet}}{Q} \qquad \ldots(10)$$

where 'R' is the thermal resistance of the heat sink geometry which is measured in K/W.

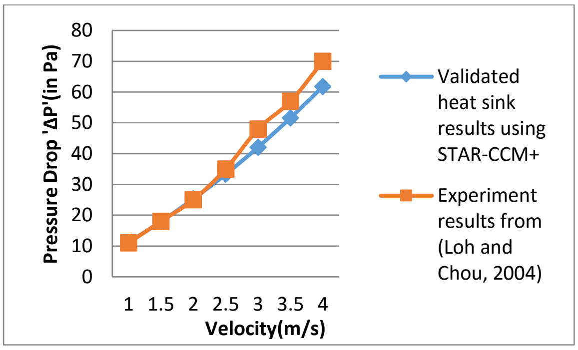
4.0 Validation Results

The previous published study conducted by (Loh and Chou, 2004) on the "comparative analysis of heat sink pressure drop using different methodologies" was used for validation [10]. The work done in their study is based upon to comparison between the results obtained by using theoretical, experimental and numerical study. In this study, the results was created with CFD tool STAR-CCM+ software and trying to match with the validation case experiment results.

Table 7: Tabular Form of Comparison of results between Validated Heat Sink and Experimental Results from (Loh and Chou, 2004)

Velocity (In M/S)	Pressure (In Pa)		Pressure Drop (ΔP)	Experimental Pressure Drop (ΔP_{exp})	% Difference In Results
	Inlet Pressure (In Pa)	Outlet Pressure (In Pa)			
1.0	101324.1	101312.9	11.2	11	1.81%
1.5	101323.3	101305.4	17.9	18	0.55%
2.0	101322.2	101296.9	25.3	25	1.20%
2.5	101320.8	101287.5	33.3	35	4.85%
3.0	101319.1	101277.0	42.1	48	12.29%
3.5	101317.1	101265.5	51.6	57	10.46%
4.0	101314.7	101252.9	61.8	70	11.71%

Figure 4: Comparison of Results between Validated Heat Sink and Experimental Results from (Loh and Chou, 2004).



From the results obtained from the validated heat sinks, it is found that the results obtained from both studies are in close contact with each other, so it makes a strong validation results. The increase in difference after 2.5m/s value of velocity may be due to the turbulent nature of the wind or air. The mean absolute percentage error between the studies is 6.12%. The further study was conducted at only 1m/s of velocity value where the difference is just 1.81%. At velocity value 1m/s, there is no big difference in the values of pressure drop, which provide good validity of the study.

5.0 CFD Simulation Results of Plate Heat Sink (PHS) with sloped Geometries

Traditional plate heat sink is simulated to find the results of different performance parameters which are used to compare with the other geometries of the plate heat sink. The traditional plate heat sink is named as 0° slope plate heat sink and is the general model for calculating the results for all the geometries in all approaches. Other geometries in this approach is made by extrude cutting from the side with an increment of 2.5° upto the 15° without changing the mass of plate heat sink. To control the mass of the heat sink, height of the fins vary according to the requirement.

5.1 Temperature profile results

Figure 5: Temperature Profile of Traditional Plate Heat sink (TPHS) or 0° slope PH

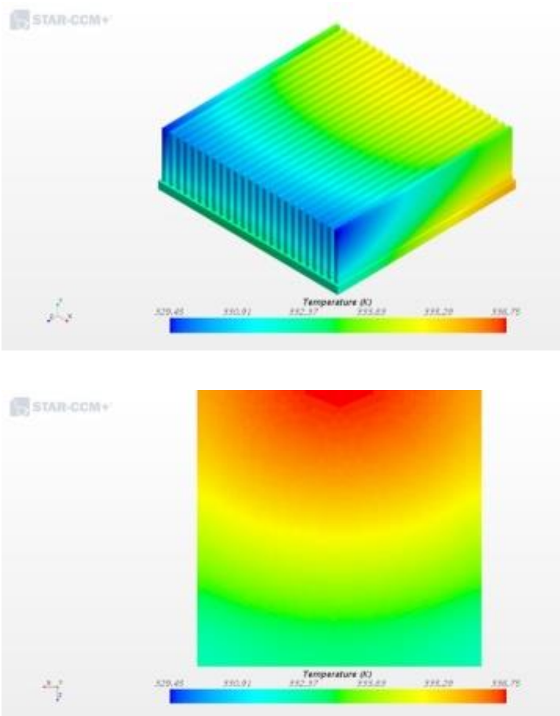
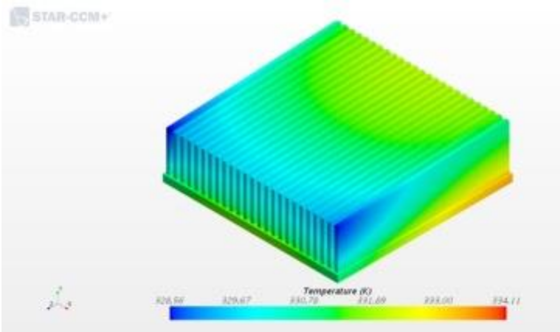


Figure 6: Temperature Profile of Plate Heat Sink (PHS) with 2.5 Slope



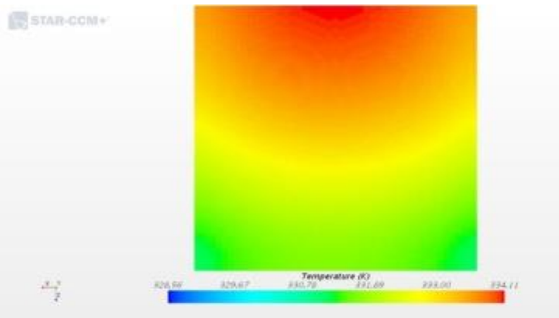


Figure 7: Temperature Profile of Plate Heat Sink (PHS) with 5° Slope

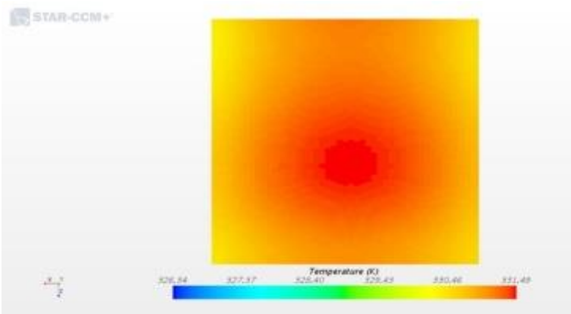


Figure 9: Temperature Profile of Plate Heat Sink (PHS) with 10° Slope

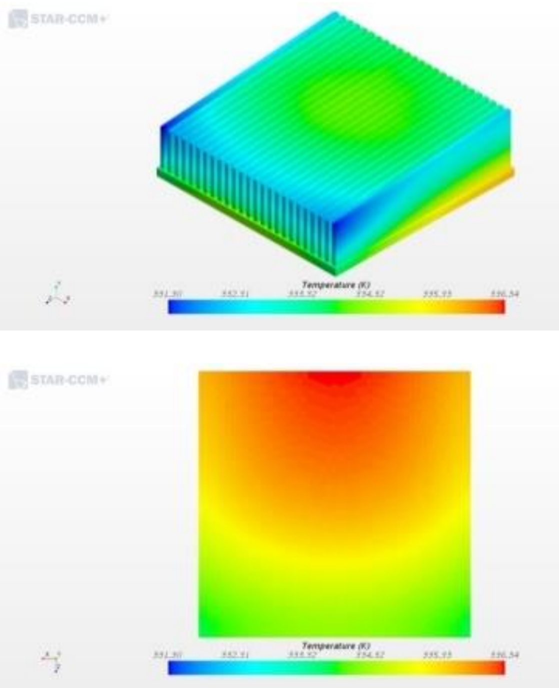


Figure 8: Temperature Profile of Plate Heat Sink (PHS) with 7.5° Slope

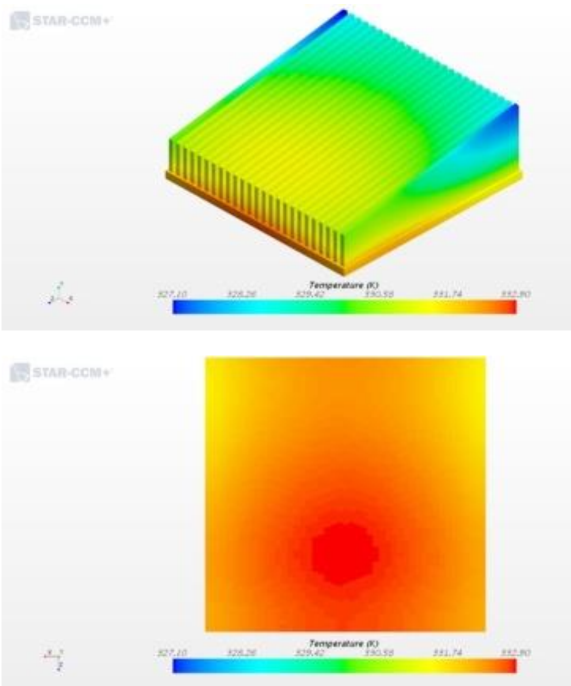
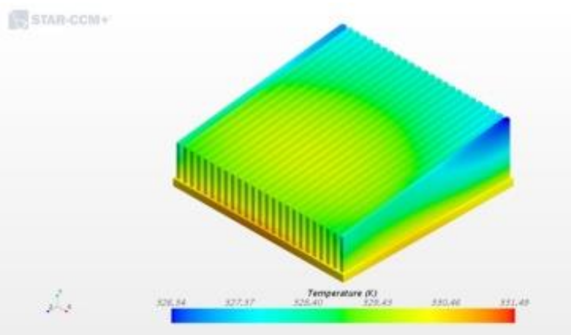
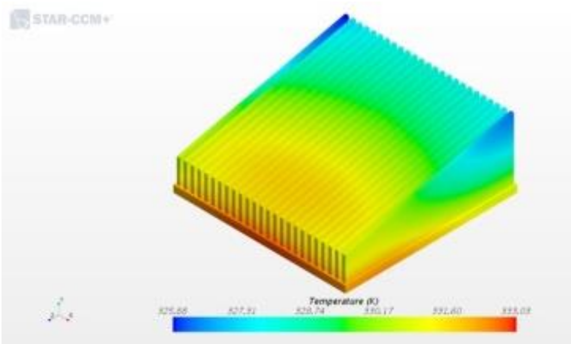


Figure 10: Temperature Profile of Plate Heat Sink (PHS) with 12.5° Slope





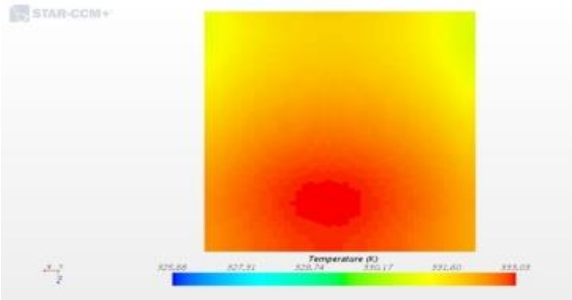
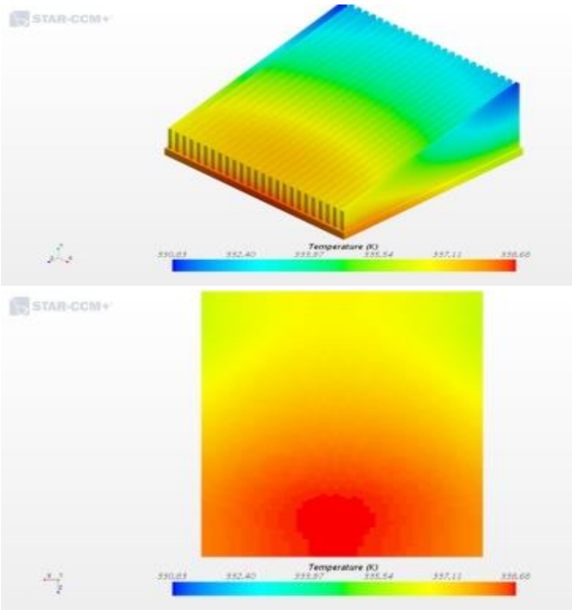


Figure 11: Temperature Profile of Plate Heat Sink (PHS) with 15° Slope



The isometric and the bottom view of temperature profiles of the 7 model geometries of plate heat sink (PHS) with sloped geometries are shown in figure 5.

5.2 Performance parameter results

Table 8: Performance Parameter Results of Plate Heat Sink (PHS) with Sloped Geometries

Type	Temperature		Pressure		Thermal Resistance	
Slope (in degrees)	T_{max} (in Kelvin)	% increase from TPHS	ΔP (in Pascal)	% increase from TPHS	R_T (in K/W)	% increase from TPHS
0.0°	**336.7**	**0%**	**28.3**	**0%**	**0.367**	**0%**
2.5°	334.1	0.77%	15.2	46.28%	0.341	7.08%
5.0°	336.3	0.11%	17.1	39.57%	0.363	1.08%
7.5°	331.4	1.57%	14.7	48.05%	0.314	16.87%
10°	332.9	1.12%	9.8	65.37%	0.329	10.35%
12.5°	333.0	1.09%	11.9	57.95%	0.330	10.08%
15°	338.6	-0.56%	11.3	60.07%	0.386	-5.17%

Figure 12: Max Temp. vs Slope Angle Graph of Plate Heat Sink (PHS) with Sloped Geometry

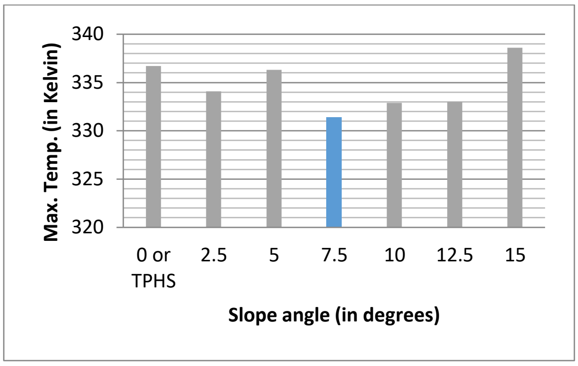


Figure 13: Pressure Drop vs Slope Angle Graph of Plate Heat Sink (PHS) with Sloped Geometries

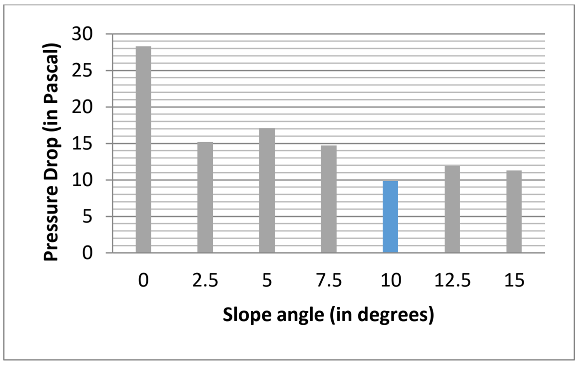
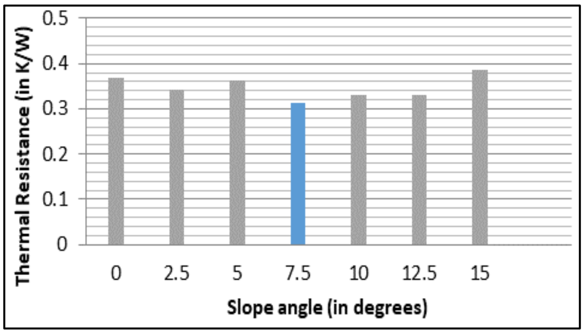


Figure 14: Thermal Resistance vs Slope Angle Graph of Plate Heat Sink (PHS) with Sloped Geometries



6.0 Conclusions

From the results, maximum temperature and thermal resistance performance of 7.5° slope geometry is increased by 1.57% and 16.87% respectively and pressure drop performance of 10° slope geometry is increased by 65.37%. Another considerable factor is the temperature profile at the bottom of the sloped heat sink geometries which is more uniformly centered in case of 7.5° and 10° slope. The conclusion stated that the angular sloped geometries provide the better result than the TPHS (Traditional Plate Heat Sink) geometry.

Reference

[1] B Freegah, AA Hussain, AH Falih, H Towsyfyan. CFD analysis of heat transfer enhancement in plate-fin heat sinks with fillet profile: investigation of new designs, Thermal Science and Engineering Progress, 2019.

[2] S Lee, M Early, M Pellilo. Thermal interface material performance in microelectronics packaging applications, Microelectronics, 28(1), 1997, 13-20.

[3] B Xia, DW Sun. Applications of computational fluid dynamics (CFD) in the food industry: a review, Computers and Electronics in Agriculture, 34, 2002, 5-24.

[4] S Subramanyam, KE Crowe. Rapid design of heat sinks for electronic cooling using computational and experimental tools, Institute of Electrical and Electronics Engineers, 16, 2000, 243-251.

[5] R Arularasan, R Velraj. CFD analysis in a heat sink for cooling electronic devices, International Journal of the computer, the internet and management, 6, 2008, 1-11.

[6] H Sivasankaran, G Asirvatham, J Bose, B Albert. Experimental analysis of parallel plate and cross cut pin fin heat sinks for electronics cooling applications, Thermal Science, 14(1), 2010, 147-156.

[7] P Rodgers, V Eveloy. Air cooled heat sink design optimization in free convection, 29th IEEE Semiconductor Thermal Measurement and Management Symposium, 2013, 170-172.

[8] D Gupta, V Venkataraman, R Nimje. CFD & Thermal Analysis of Heat Sink and its Application in CPU", International Journal of Emerging Technology and Advanced Engineering, 4(8), 2014, 198-202.

[9] SS Haghighi, HR Goshayeshi, MR Safaei. Natural convection heat transfer enhancement in new designs of plate-fin based heat sinks, International Journal of Heat and Mass Transfer, 125, 2018, 640-647.

[10] CK Loh, DJ Chou. Comparative analysis of heat sink pressure drop using different methodologies, Twentieth Annual IEEE Semiconductor Thermal Measurement and Management Symposium, 2004, 6