Study of Active Plate Heat Sink using Different Slope Angles

[1]Department of Mechanical Engineering, IK Gujral Punjab Technical University, Jalandhar, Punjab, India.
[2]Department of Mechanical Engineering, Khalsa College of Engineering & Technology, Amritsar, Punjab, India.

Article Info
Article history:
Received 29 July 2020
Received in revised form
20 October 2020
Accepted 28 October 2020
Available online 15 December 2020

Keywords: Computational Fluid Dynamics, Active heat sink, performance parameters, slope angle,

Abstract:

1. Introduction

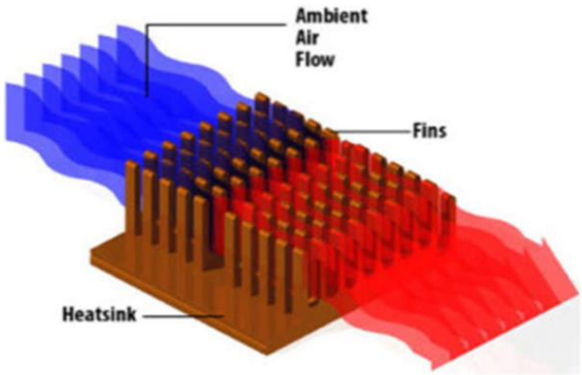

Fig. 1:

2. Literature survey

Corresponding Author,
E-mail address: amanbirsingh1992@gmail.com;
Phone No--+91- 8872344739

3. Methodology

Table 1:

Design Notation	Dimensions	Condition

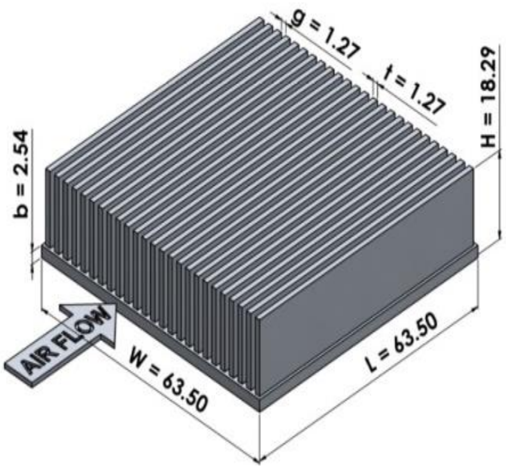

Fig. 2:

3.1 CAD Modeling

-
-

3.2 CFD Simulation

3.2.1. Pre-processing

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

3.2.3 Meshing

Table 2:

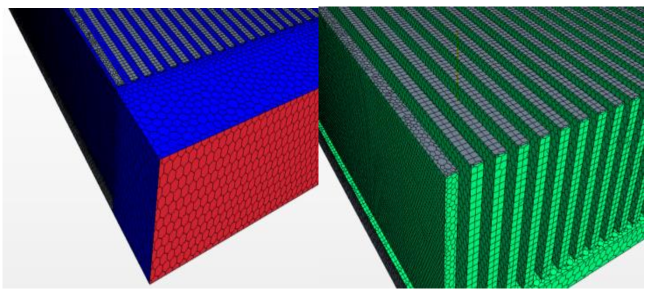

Fig. 3:

3.3 Physics models used for fluid and solid

Table 3:

Fluid	Comments

Table 4:

Solid	Comments

3.4 Reynolds number

$$\frac{\rho v D}{\mu} \quad (1)$$

$$D = \frac{4A}{P} = 0.025\text{m} \quad (2)$$

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

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

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,

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

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

The momentum equation in three-dimensions is given by,

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

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

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

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,

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

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,

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

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.

- 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,

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

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

4. 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 ($\Delta P_{exp.}$)	% Difference In Results
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%

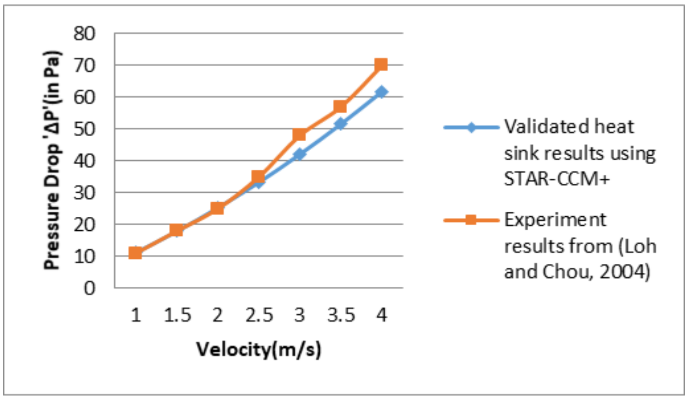

Fig. 4: Comparison of results between validated heat sink and experimental results from (Loh and Chou, 2004).

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. 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

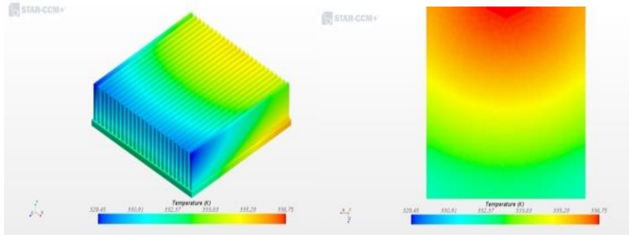

Fig. 5: Temperature profile of traditional plate heat sink (TPHS) or 0° slope PH

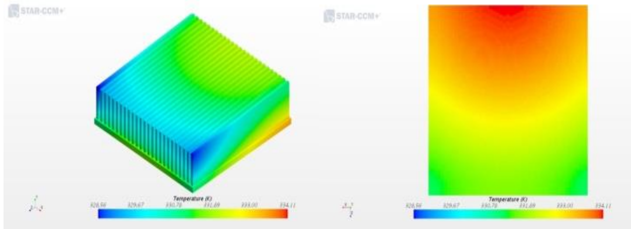

Fig. 6:

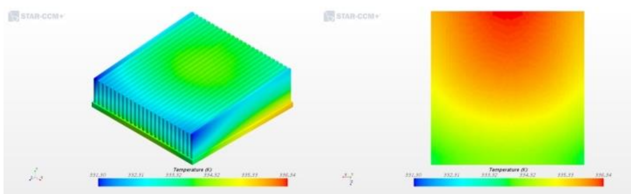

Fig. 7:

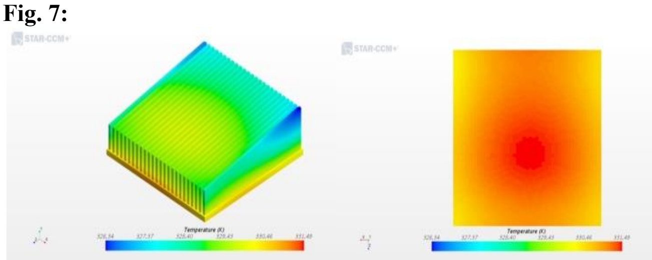

Fig. 8:

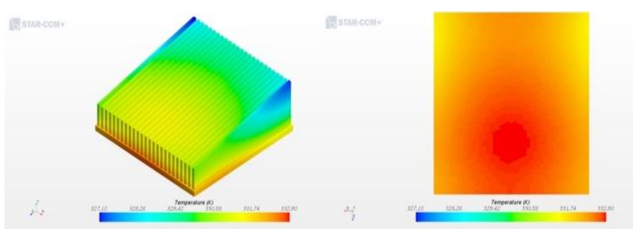

Fig. 9:

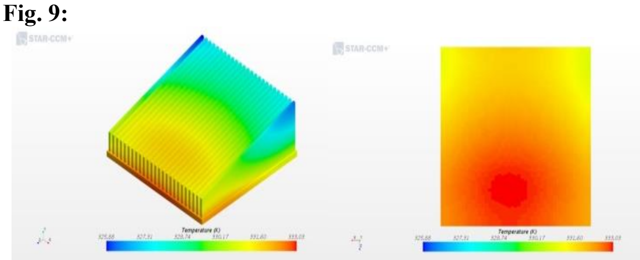

Fig. 10:

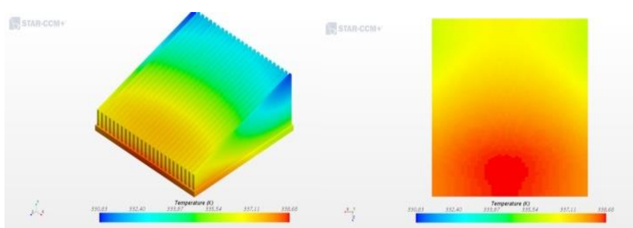

Fig. 11:

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:

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%**

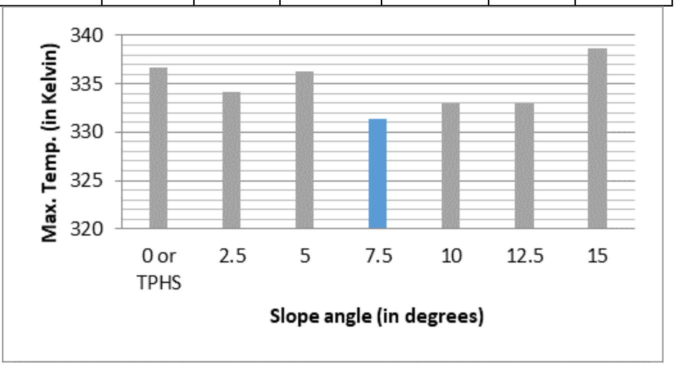

Fig. 12: Max. Temp. vs Slope angle graph of plate heat sink (PHS) with sloped geometry

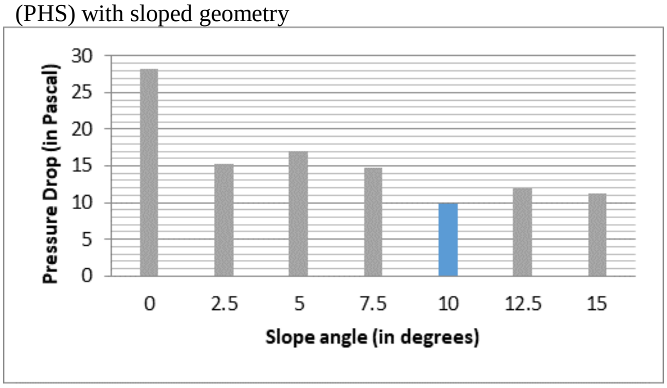

Fig. 13:

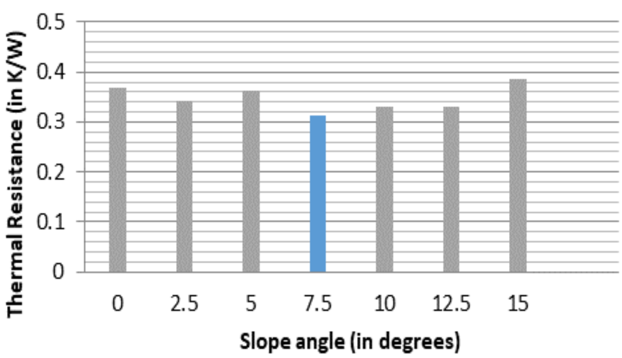

Fig. 14:

6. 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