JETIR.ORG ISSN: 2349-5162 | ESTD Year : 2014 | Monthly Issue
JOURNAL OF EMERGING TECHNOLOGIES AND INNOVATIVE RESEARCH (JETIR)
An International Scholarly Open Access, Peer-reviewed, Refereed Journal

AERODYNAMIC AND HEAT DISSIPATION ANALYSIS OF HEAT SINK DESIGN IN COOLING OF CPU

by
PRASHANT DAHIYA
20600396
2020PCD6201

NETAJI SUBHAS UNIVERSITY OF TECHNOLOGY
(Formerly Netaji Subhas Institute of Technology)
Azad Hind Fauj Marg, Sector-3, Dwarka, New Delhi-110078

ABSTRACT

In today's day and age with the increase in CPU performance better heat ventilation is required to cool down the CPU which helps it to work more efficiently, according to Moore's Law the number of transistors on a micro chip doubles every 2 year, with the decrease in size, processor chip produces more heat so better cooling mechanisms are needed to be incorporated for better cooling of the CPU, as we are aware of the fact that CPU performance decreases with the increase in temperature, thereby reducing it's lifespan. In this research paper we have carried out our analysis in SOLIDWORKS software, in order to improve the efficiency of airflow by twisting the heat sink blades for better air mixing and ventilation.

1 INTRODUCTION

A heat sink (also commonly spelled heatsink) is a passive heat exchanger that transfers the heat generated by an electronic or a mechanical device to a fluid medium, often air or a liquid coolant, where it is dissipated away from the device, thereby allowing regulation of the device's temperature.

Heat distributes throughout the heat sink. Heat will naturally travel through the heat sink via natural conduction moving across the thermal gradient from a high temperature to a low-temperature environment.

Heat Sinks are of majorly of 2 types:

A) Passive: It rely on natural convection, meaning no external cooling source isattached to it like a fan or a liquid medium.

B) Active: It rely on forced convection, meaning some external cooling source isattached to it like a fan or a liquid medium.

In our study we have carried out different configuration of radial heat sink and tested the cooling efficiency of each configuration in order to determine the best heat sink using SOLIDWORKS software.

2 LITERATURE REVIEW

Literature review has been done on the most recent developments in the areas of heat sink. Most of the research that has been carried out does not focus on the aerodynamics and overall design details of the heat sink, in our research we have mainly focused on the design of our heat sink and the effect of split heat sink over straight, analysis results shows better cooling effects due to better design of overall heat sink.

[1] and [2] helps us understand the basic concepts for carrying out our research. [3] The computational fluid dynamics is concentrated on the forced air cooling of the CPU using a heat sink. This paper utilizes CFD to identify a cooling solution for a desktop computer, which uses an 80 W CPU. [4] The correlation predicts the Nusselt number and the dimensionless pressure drop and takes into account the influence of duct height, duct width, fin height, fin thickness, and fin-to-fin distance. The correlation parameters are specific for each fin type. [5] The thermal performance of rectangular plate fin, staggered fin, circular pin fin and elliptical pin fin heat sinks are compared for the fixed base plate dimensions and fin height under fixed volume conditions. [6] takes into consideration various heat intensive environmental conditions based on size and geometry of our fin. [7] discusses various cooling techniques based on heat generation of electric devices. [8] they compared thermal performances of plate-fin and pin-fin heat sinks subject to an impinging flow, a model based on the averaging method for predicting the pressure drop and the thermal resistance. [9] talks about For constant fin height, the fin width that provides the best thermal performance increases with the Reynolds number. Further increasing the fin width will degrade the thermal performance. [10] a 50 percent reduction in the heat sink performance. A correlation which relates the heat sink flow rate to the approach flow rate and heat sink fin density has been developed which quantifies the bypass effect for straight rectangular finned heat sinks. [11] The purpose of this study is to investigate the effects of lateral perforation shapes on the thermal performance of heat sinks in comparison with the regular solid fins under turbulent flow condition. [12] The correlation predicts the Nusselt number and the dimensionless pressure drop and takes into account the influence of duct height, duct width, fin height, fin thickness, and fin-to-fin distance. The effects of spacing, length, thickness, and material of fifins on the natural convection heat transfer of fin arrays were systematically studied by Leung and his co-workers [12–21]. The optimized vertical fin array parameters as a function of dimensions, thermal conductivity, emissivity, and fluid properties were numerically examined by Lieto Vollaro et al. [22]. Optimization of horizontal fin arrays was also conducted by Baskaya et al. [23] with a finite volume model. The orientation of rectangular fin heat sink has a strong effect on the heat transfer performance under natural convection conditions. Starner and McManus [24] experimentally studied the natural convection of heated rectangular fifin arrays in three orientations, i.e. 0 , 45 , and 90 . Leung and Probert [25] conducted studies in ori entations of 0 , 90 and 270 (defined in Fig. 1(c)). Dayan and his co workers [26,27] performed researches into downward facing fifin arrays with inclinations within 30 . It was suggested to tilt the array to angles greater than 10 to prevent flow separation and achieve a higher heat transfer rate. Tari and Mehrtash [28] conducted numerical researches on rectangular fin arrays in several inclinations from the vertical fin array, and found that small inclinations did not reuce heat transfer rate.

3 EQUATION USED

- Newton Law of Cooling dQ = hA∇T

  h - convective heat transfer coefficient

  ∇T - change in temperature A - surface area dQ - change in heat

- Fourier's Law of Heat Conduction

  $$Q = -kA\frac{dt}{dx}$$

  Q - Heat flow through a body per unit time

  k - thermal conductivity A - surface area of heat flow dt - change in temperature in kelvin or Celsius dx - thickness of body

- General Heat Conduction Equation

  $$\frac{}{k} = \frac{1}{\alpha} \cdot \frac{□□}{□□} \quad \nabla 2t + q_g$$

  $$\frac{}{\rho c} \qquad k\ \alpha =$$

  $$\nabla 2t = \partial x \partial 22t + \partial y \partial 22t + \partial z \partial 22t$$

  $\nabla^2 t$ - Laplace Equation

  □□ - Heat Generated per unit volume per unit time k - thermal conductivity α - thermal diffusivity c - specific heat capacity

- Heat dissipation from Fin

  □□□□ = □□□□□ hP m = $\sqrt{kA}$

  Q_{fin} = Heat dissipated by the fin k - thermal conductivity

  A - area of cross section

  ∇T - change in temperature through the fin P - perimeter of cross section of the fin

- Nusselt Number (Nu)

  $$Nu = \frac{Q_{conv}}{Q_{cond}} = \frac{h}{k/L} = \frac{hL}{k}$$

  h - convective heat transfer k - thermal conductivity L - length

- Reynolds Number (Re)

  $$□_□ = \frac{□□□}{\mu} = \frac{□□}{}$$

  $$□ \quad □\ \vartheta = \frac{\mu}{\rho}$$

  ϑ = kinematic viscosity μ = dynamic viscosity

  ρ = density

  V = velocity of fluid

  D = length/diameter of cross section

- Euler Equation of Motion

  $$\frac{□□}{} + □□□ + □□□ = 0$$

dp - pressure difference ρ - density g - acceleration due to gravity

z - head v - velocity Bernoulli's Equation

$$\frac{p}{\rho g} + \frac{v^2}{2g} + z = \text{constant}$$

$\frac{p}{\rho g}$ - pressure energy

$\frac{v^2}{2g}$ - kinetic energy

z - potential energy

Velocity Distribution

$$u = -\frac{1}{4\mu}\left(\frac{\partial P}{\partial x}\right)[R^2 - r^2]$$

Hagen Poiseuille Pressure Difference Formula

$$\frac{(p_1 - p_2)}{\rho g} = h_f = \frac{128\mu QL}{\rho g \pi D^4} = \frac{32\mu u L}{\rho g D^2} = \frac{8\mu u L}{\rho g R^2}$$

Q = A*V

Q - discharge u - velocity h - head

p - pressure D - diameter R - radius

g - acceleration due to gravity ρ - density

Newton law of cooling is used to find out the rate of heat dissipation by heat convection, Fourier law of heat conduction and general heat conduction are used to calculate rate of heat dissipation via heat conduction, heat dissipation via fin and nusselt formula are also be used to calculate heat dissipation in solids via fin, Reynolds number and Bernoulli's equation, velocity distribution and Hagen Poiseuille formulas are used to calculate the flow properties of fluid medium which is air. Laminar flow is desired for smooth transition of air fluid medium through the heat sink fins.

4 METHODS

While carrying out the analysis, 6 main components are taken

(i) Processor Chip	(Figure 1.1 & 1.2)
(ii) Circular Straight Heat Sink	(Figure 2.1 & 2.2)
(iii) Circular Twist Heat Sink	(Figure 3.1 & 3.2)
(iv) Circular Split Straight Heat Sink	(Figure 4.1 & 4.2)
(v) Circular Split Twist Heat Sink	(Figure 5.1 & 5.2)
(vi) Static Airflow Fan 9 Blades	(Figure 6a & 6b)
(vii) Static Airflow Fan 5 Blades	(Figure 7a & 7b)

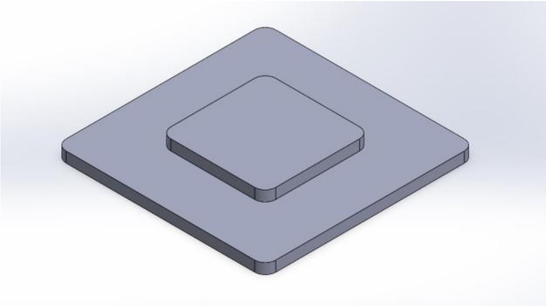

Figure 1.1
(Processor Chip Isometric View)

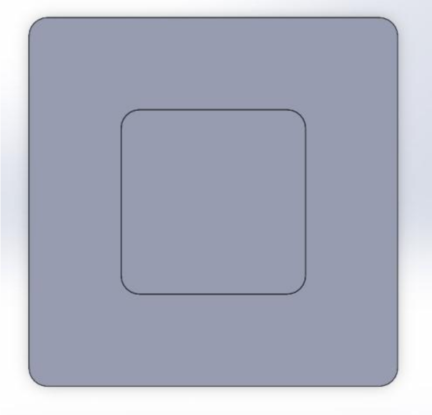

Figure 1.2
(Processor Chip Top View)

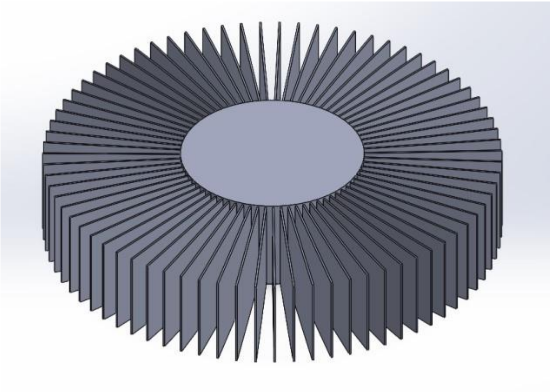

Figure 2.1
(Circular Straight Heat Sink Isometric View)

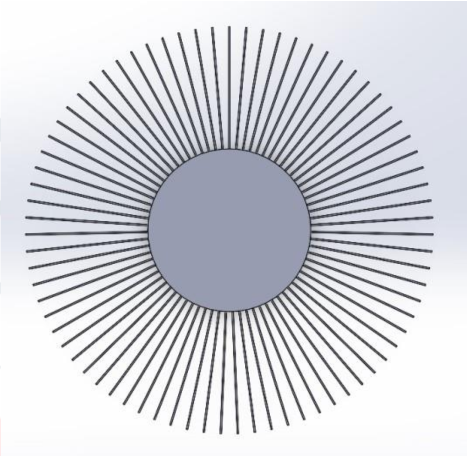

Figure 2.2
(Circular Straight Heat Sink Top View)

Figure 3.1
(Circular Twist Heat Sink Isometric View)

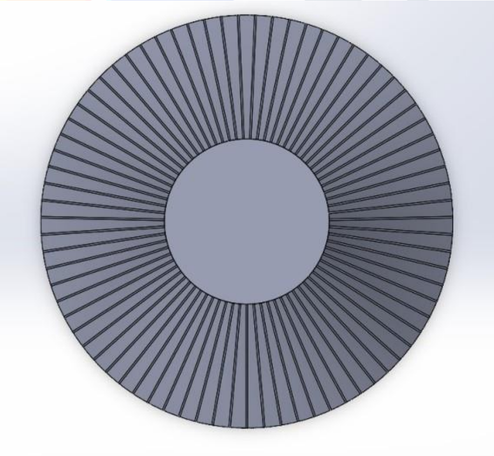

Figure 3.2
(Circular Twist Heat Sink Top View)

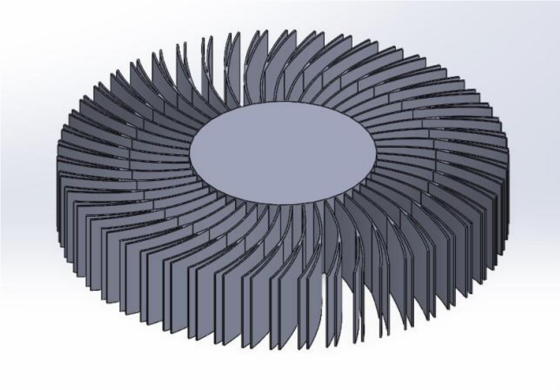

Figure 4.1
(Circular Split Straight Heat Sink Isometric View)

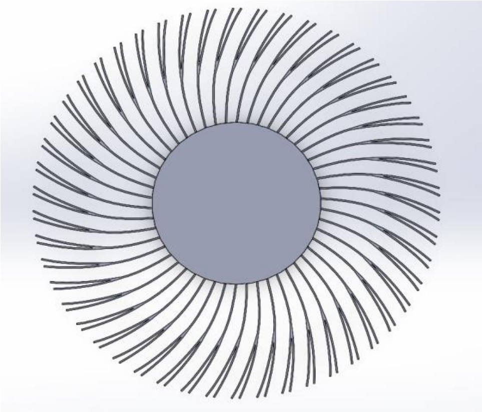

Figure 4.2
(Circular Split Straight Heat Sink Top View)

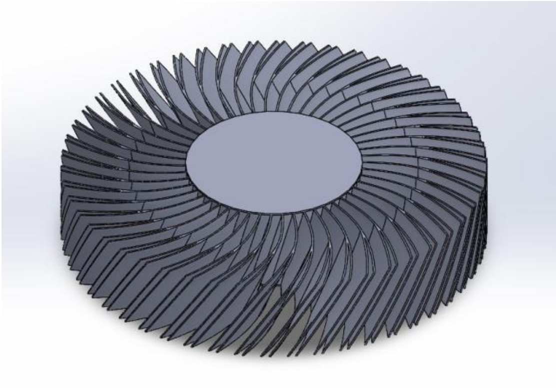

Figure 5.1
(Circular Split Twist Heat Sink Isometric View)

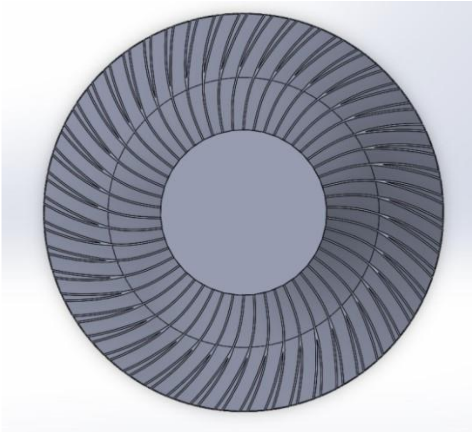

Figure 5.2
(Circular Split Twist Heat Sink Top View)

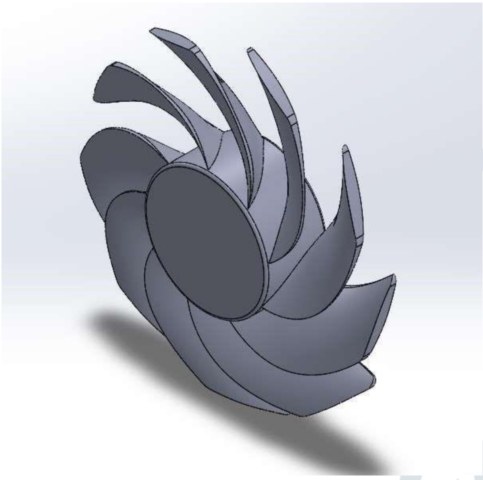

Figure 6.1
(Static Airflow Fan 9 Blades Isometric View)

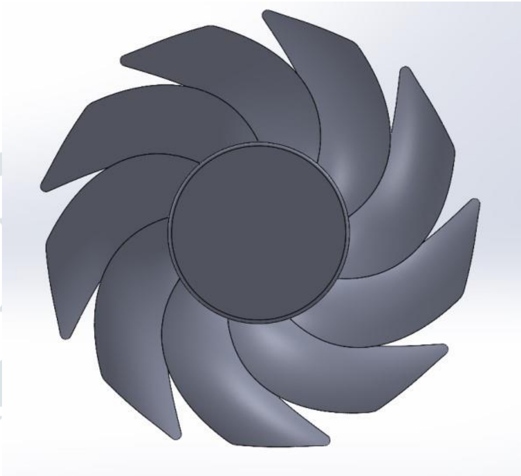

Figure 6.2
(Static Airflow Fan 9 Blades Front View)

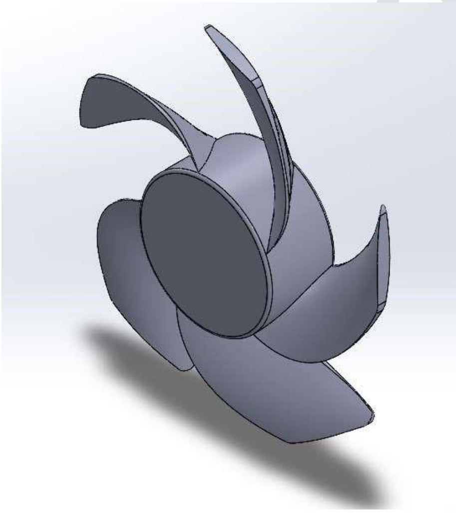

Figure 7.1
(Static Airflow Fan 5 Blades Isometric View)

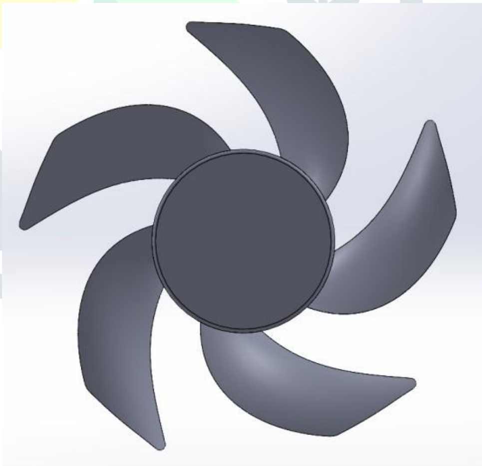

Figure 7.2
(Static Airflow Fan 5 Blades Front View)

The following illustrates the design details of our experimental work

- Processor Chip is made to be of copper, 20*20 mm is the lower base, and over that is our Processor Chip 10*10 mm, thickness 1 mm each, and fillet 1 mm all four corners. (Figure 1.1 & 1.2)
- Circular Straight is made of aluminium 1060 alloy, 10 mm diameter is our solid core and 25mm outer diameter, it has 75 fins and more number of fins are to be incorporated in Circular Straight as compared to Circular Split Straight, and it’s clearly visible that there is more density of blades leading to less heat dissipation. (Figure 2.1 & 2.2)
- Circular Twist Heat Sink is also made of aluminium 1060 alloy, same dimensions, then only difference being that it is twisted by 15 degrees using Flex command in SOLIDWORKS. (Figure 3.1 & 3.2)

- Circular Split Straight Heat Sink is made of aluminium 1060 alloy, it has 50 number of fins, 10 mm diameter is our solid core, at 17 mm diameter split is there and outer diameter is 25 mm, the curve is spline, thickness is 5 mm of blades and 5.2 mm of the solid core, 0.1 mm clearance is provided at both top and bottom, and the thickness of fin is 0.05 mm. (Figure 4.1 & 4.2)
- Circular Split Twist Heat Sink is also made of aluminium 1060 alloy, same dimensions, then only difference being that it is twisted by 15 degrees using Flex command in SOLIDWORKS. (Figure 5.1 & 5.2)

5 FLOW SIMULATION AND THERMAL SIMULATION STUDY

In this study our main focus was to check the heat dissipation, and the temperature at the tip of the fin. We have performed two studies, one with circular split straight and other with circular split twist. Here are the types of simulations that were conducted

(i) Circular Straight Heat Sink Thermal Simulation (Figure 2.3 & 2.4 & 2.5)
(ii) Circular Straight Heat Sink Flow Simulation (Figure 2.6 & 2.7)
(iii) Circular Twist Heat Sink Thermal Simulation (Figure 3.3 & 3.4 & 3.5)
(iv) Circular Twist Heat Sink Flow Simulation (Figure 3.6 & 3.7)
(v) Circular Split Straight Heat Sink Thermal Simulation (Figure 4.3 & 4.4 & 4.5)
(vi) Circular Split Straight Heat Sink Flow Simulation (Figure 4.6 & 4.7)
(vii) Circular Split Twist Heat Sink Thermal Simulation (Figure 5.3 & 5.4 & 5.5)
(viii) Circular Split Twist Heat Sink Flow Simulation (Figure 5.6 & 5.7)
(ix) Static Airflow Fan 9 Blades Flow Simulation (Figure 6.1 & 6.2)
(x) Static Airflow Fan 5 Blades Flow Simulation (Figure 7.1 & 7.2)

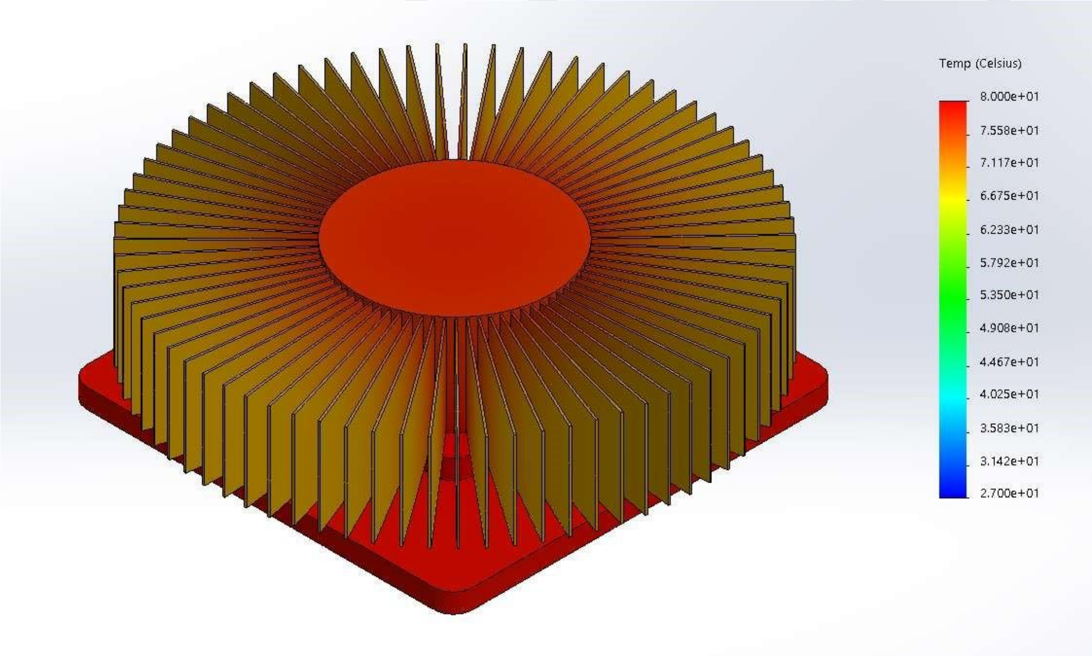

Figure 2.3
(Circular Straight Heat Sink Thermal Simulation Isometric view)

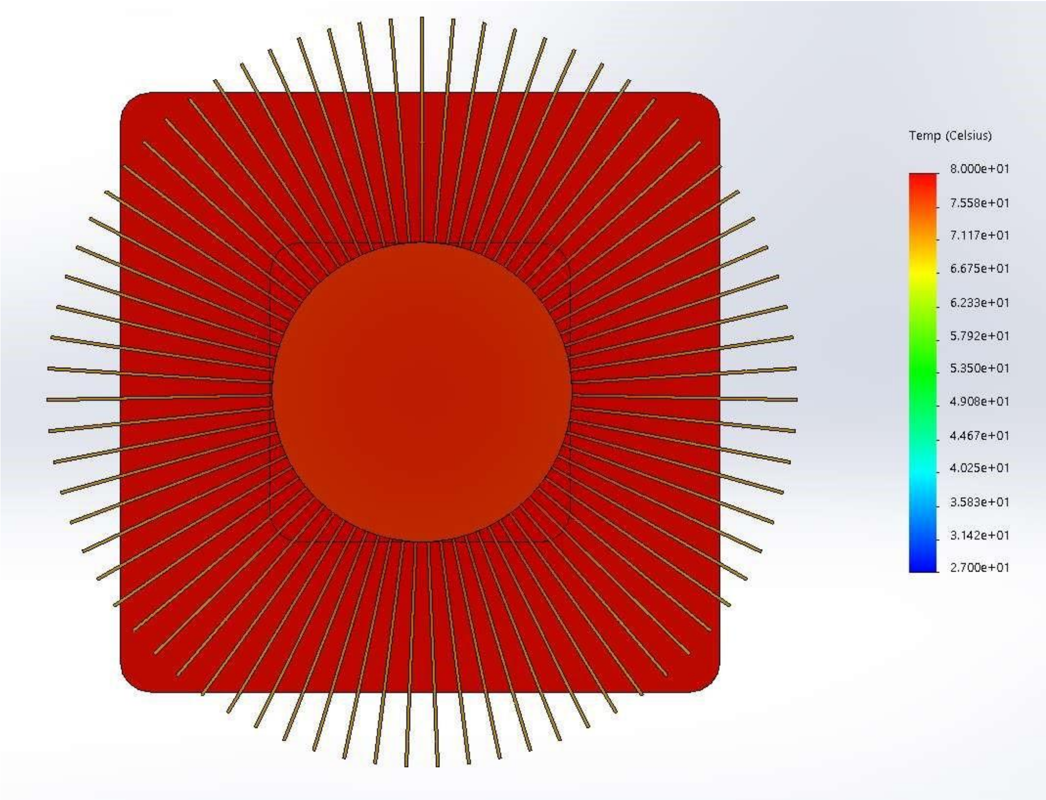

Figure 2.4
(Circular Straight Heat Sink Thermal Simulation Top view)

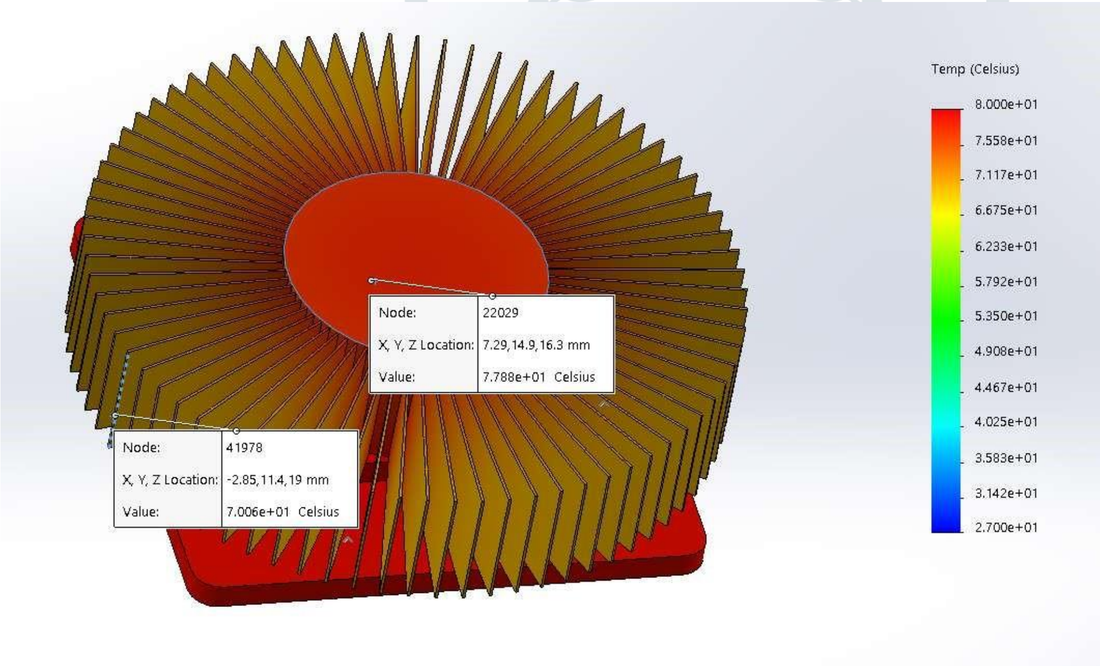

Figure 2.5
(Circular Straight Heat Sink Thermal Simulation Node Temperature at Fin Tip and center)

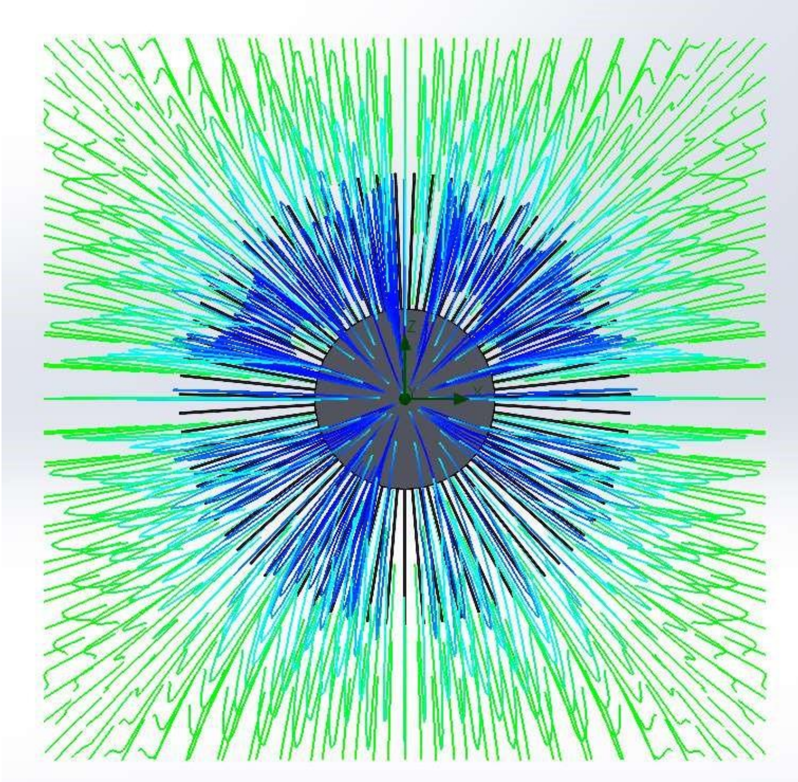

Figure 2.6
(Circular Straight Heat Sink Flow Simulation Axial View)

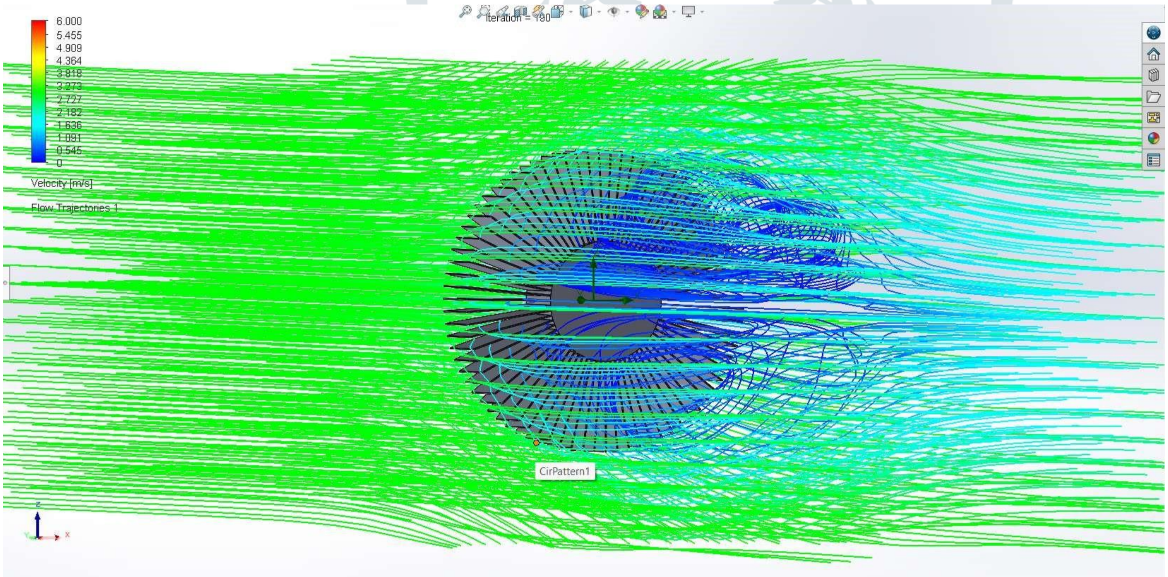

Figure 2.7
(Circular Straight Heat Sink Flow Simulation Lateral View)

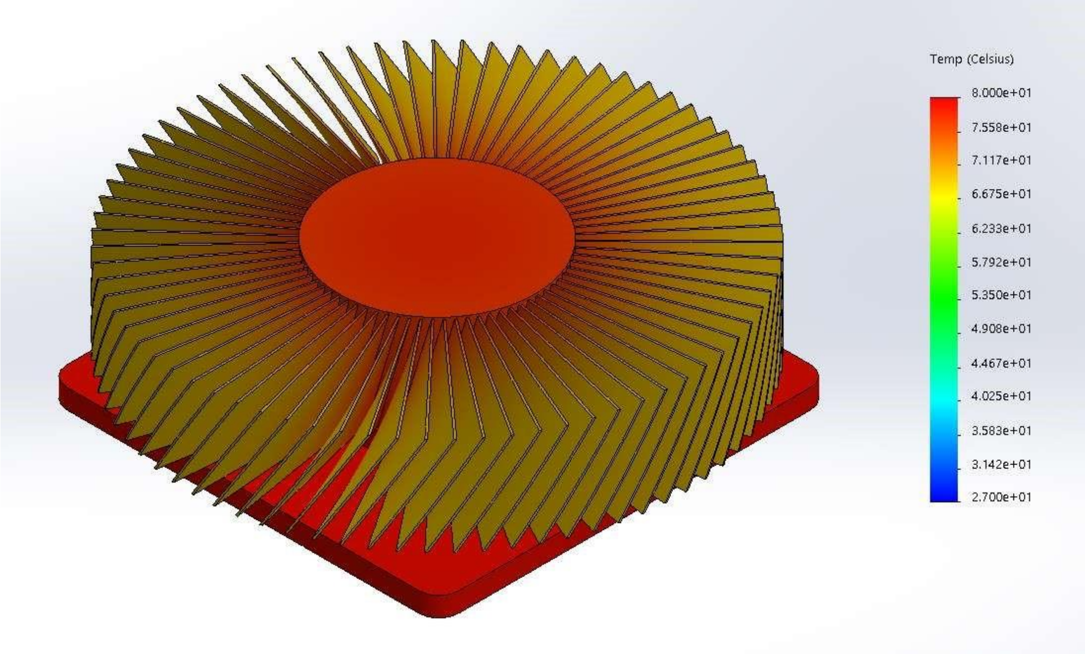

Figure 3.3
(Circular Twist Heat Sink Thermal Simulation Isometric view)

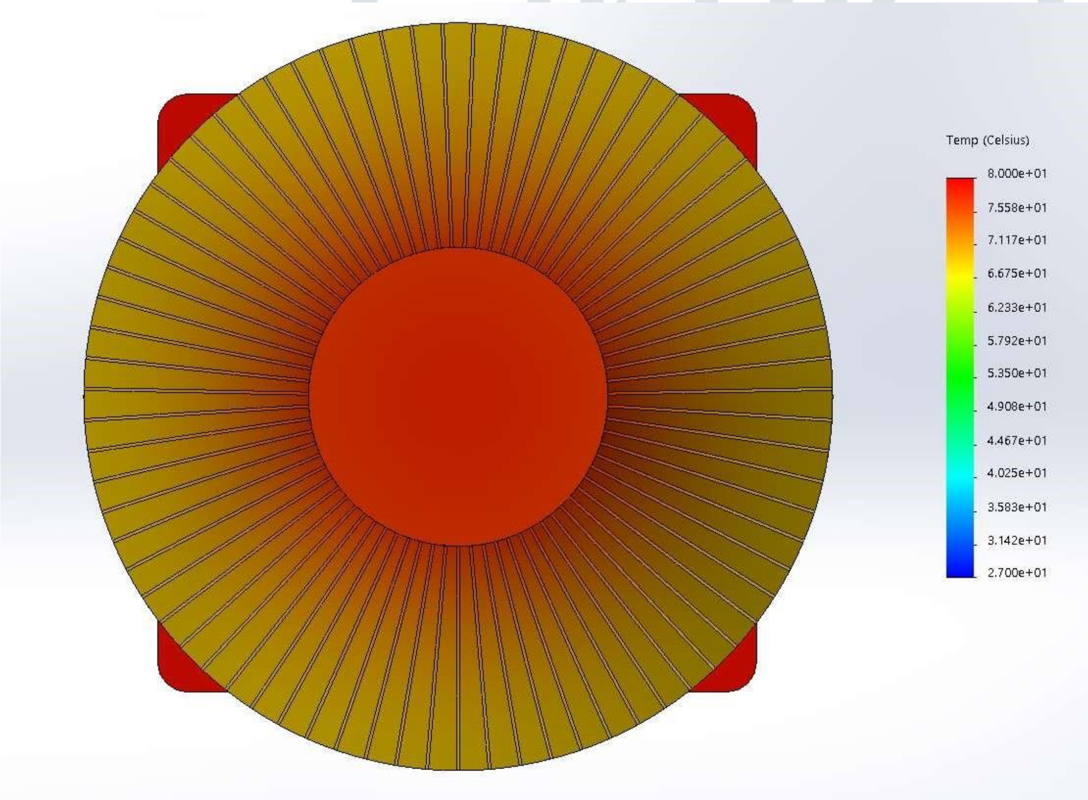

Figure 3.4
(Circular Twist Heat Sink Thermal Simulation Top view)

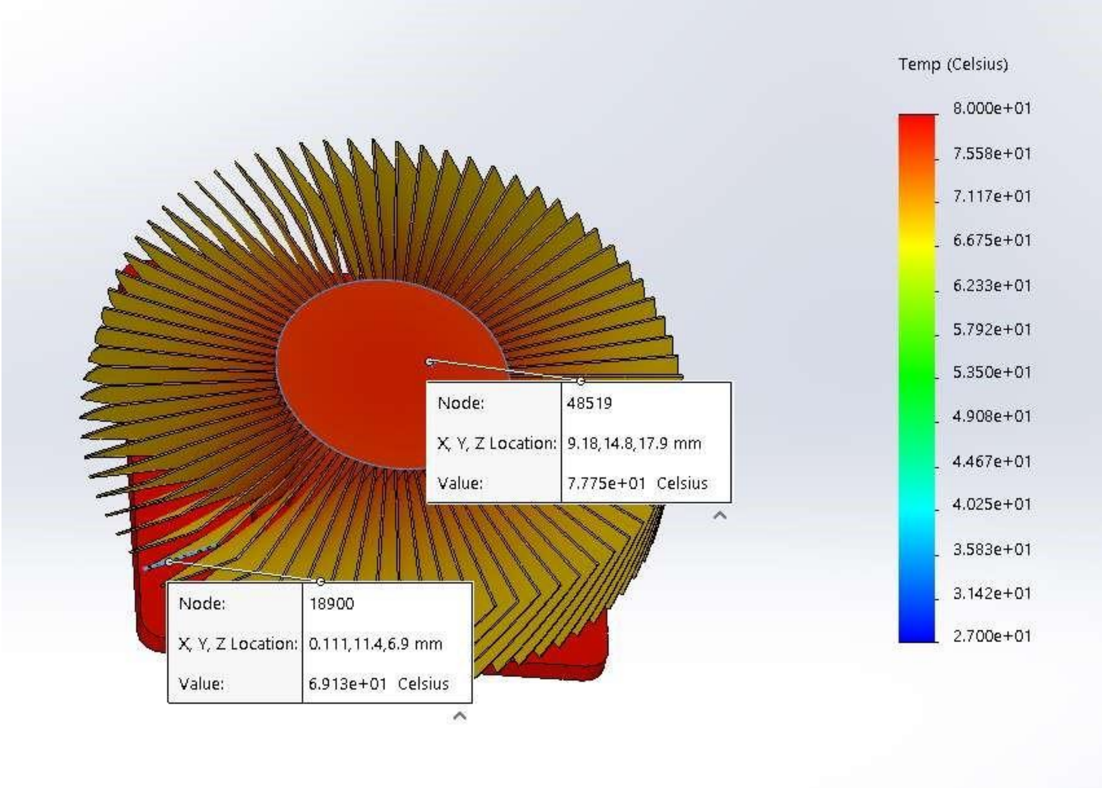

Figure 3.5
(Circular Twist Heat Sink Thermal Simulation Node Temperature at Fin Tip and center)

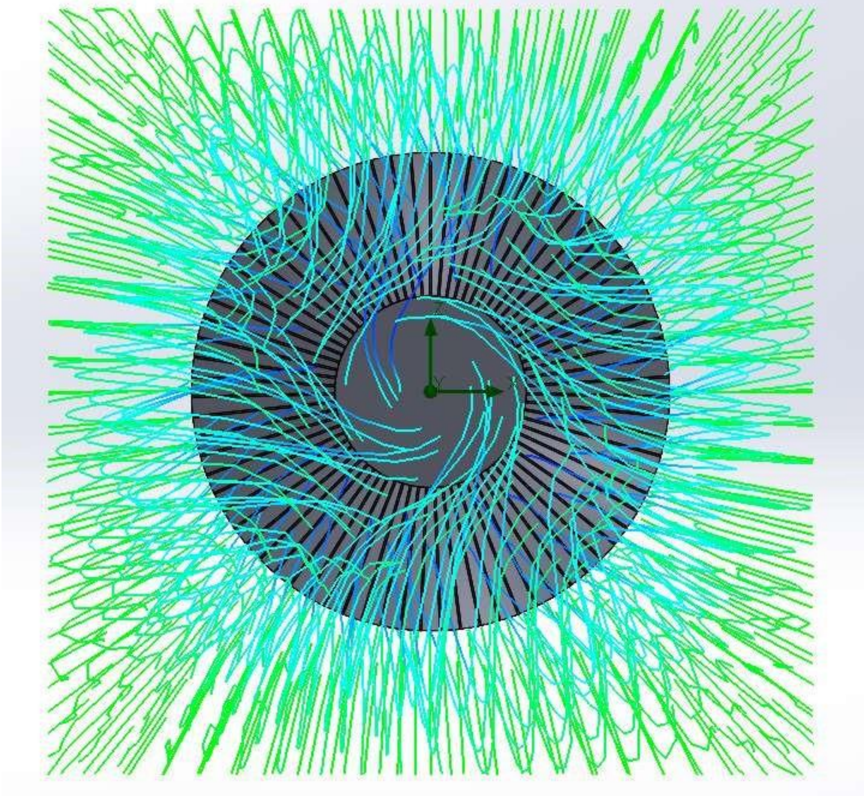

Figure 3.6
(Circular Twist Heat Sink Flow Simulation Axial View)

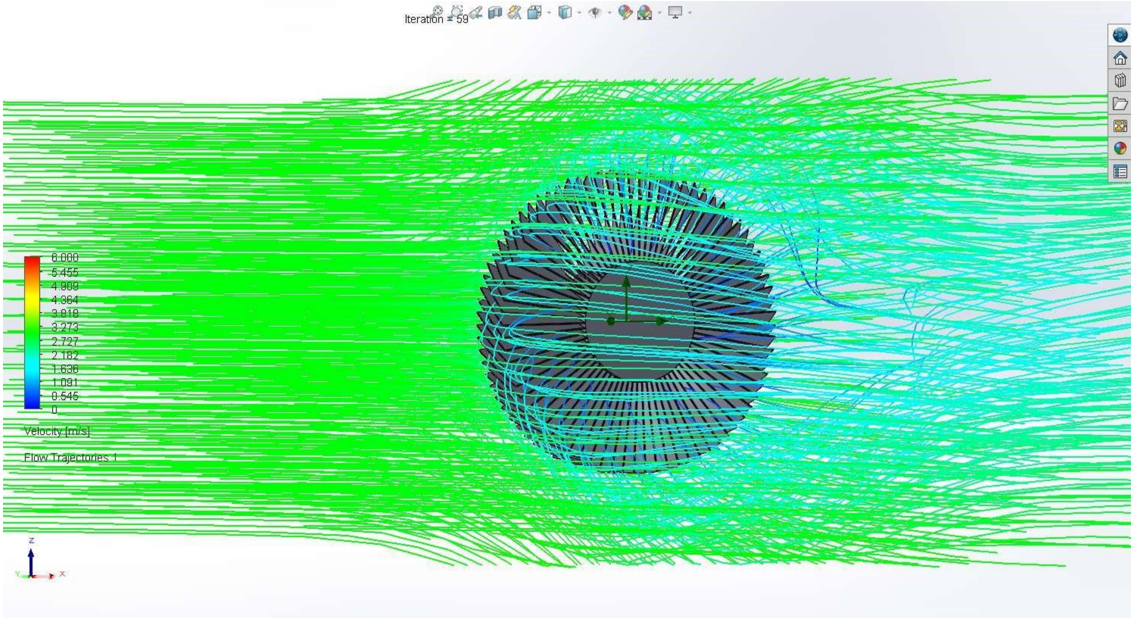

Figure 3.7
(Circular Twist Heat Sink Flow Simulation Lateral View)

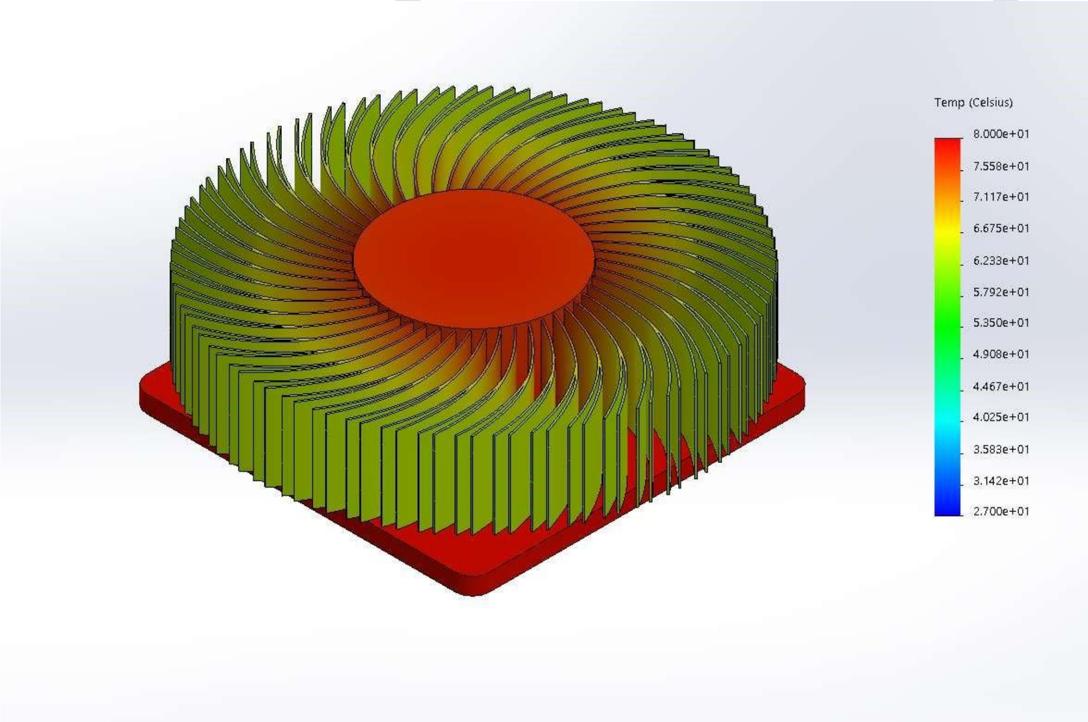

Figure 4.3
(Circular Split Straight Heat Sink Thermal Simulation Isometric view)

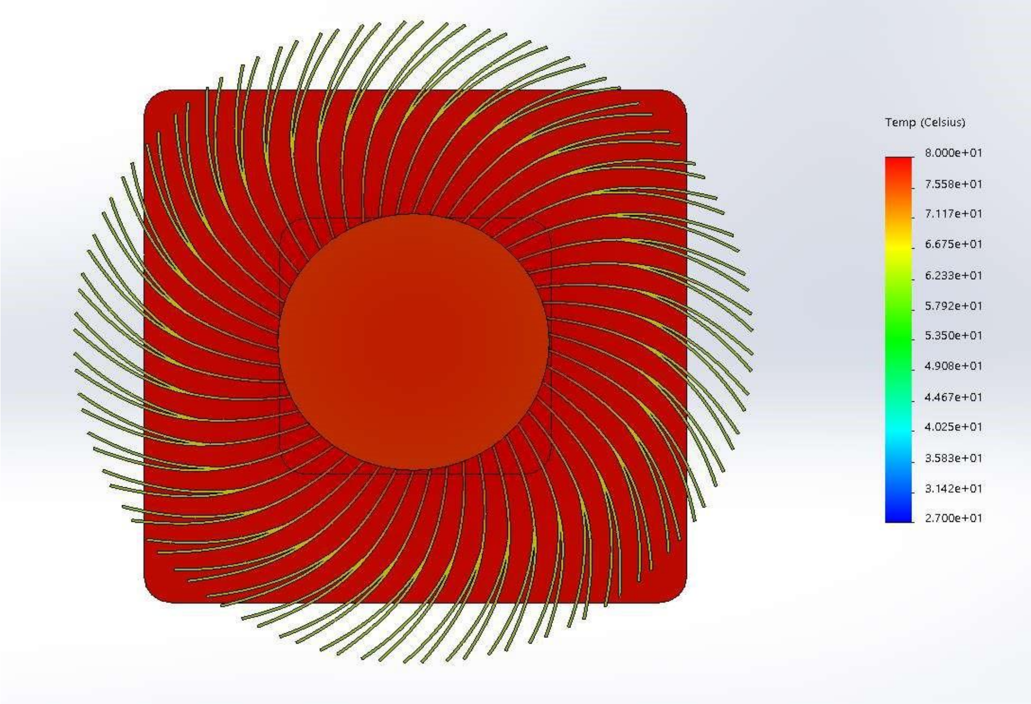

Figure 4.4
(Circular Split Straight Heat Sink Thermal Simulation Top view)

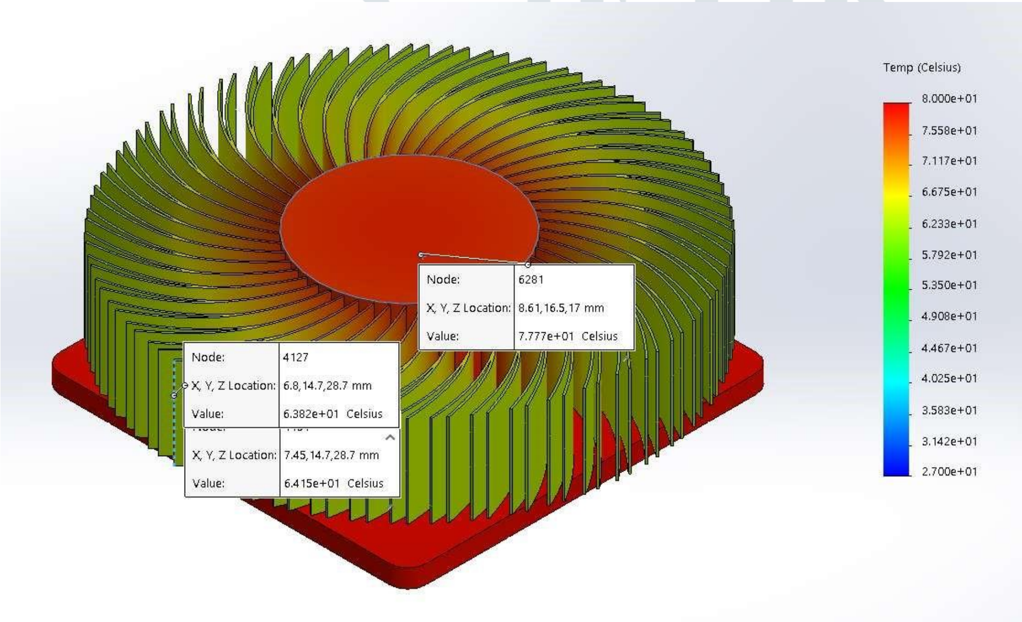

Figure 4.5
(Circular Split Straight Heat Sink Thermal Simulation Node Temperature at Fin Tip and center)

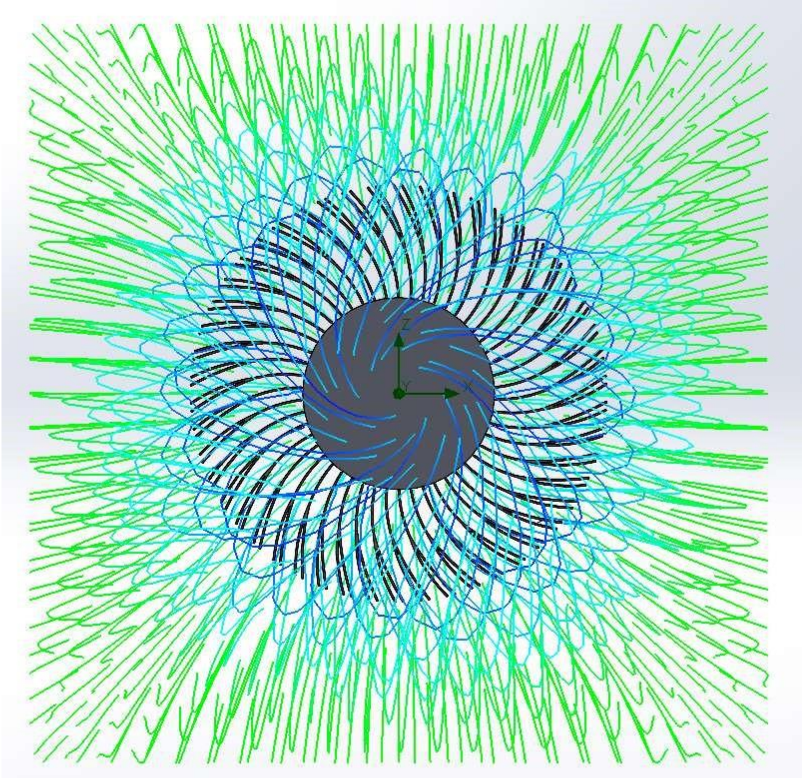

Figure 4.6
(Circular Split Straight Heat Sink Flow Simulation Axial View)

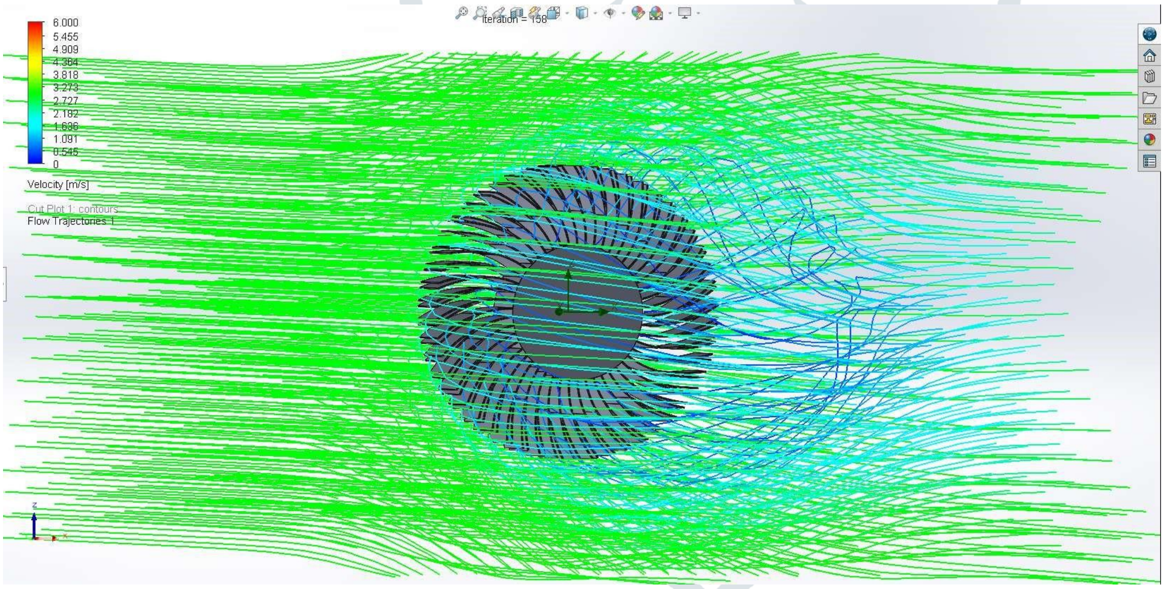

Figure 4.7
(Circular Split Straight Heat Sink Flow Simulation Lateral View)

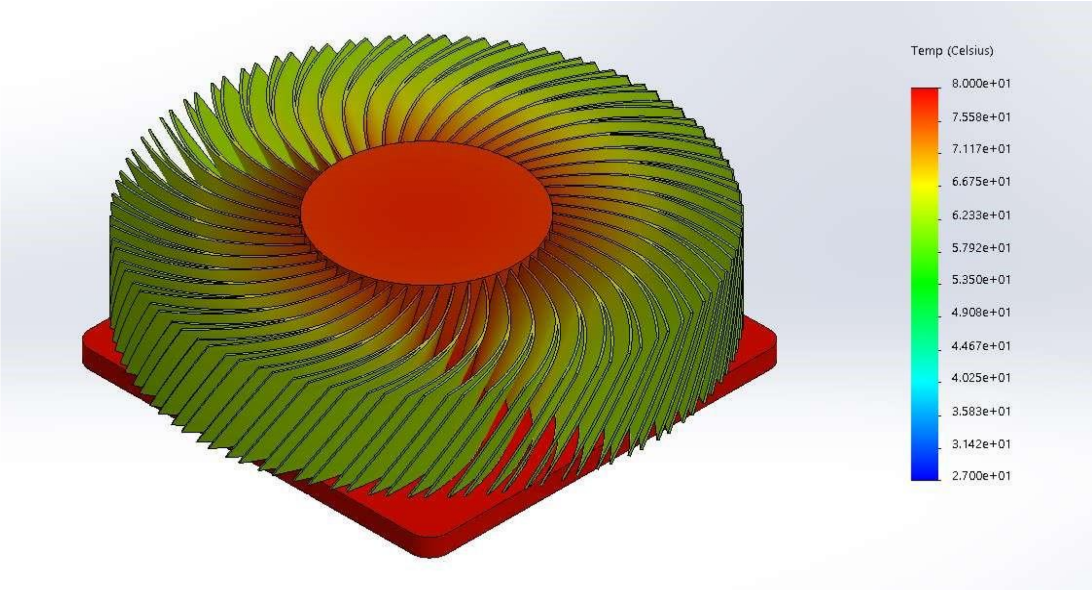

Figure 5.3
(Circular Split Twist Heat Sink Thermal Simulation Isometric view)

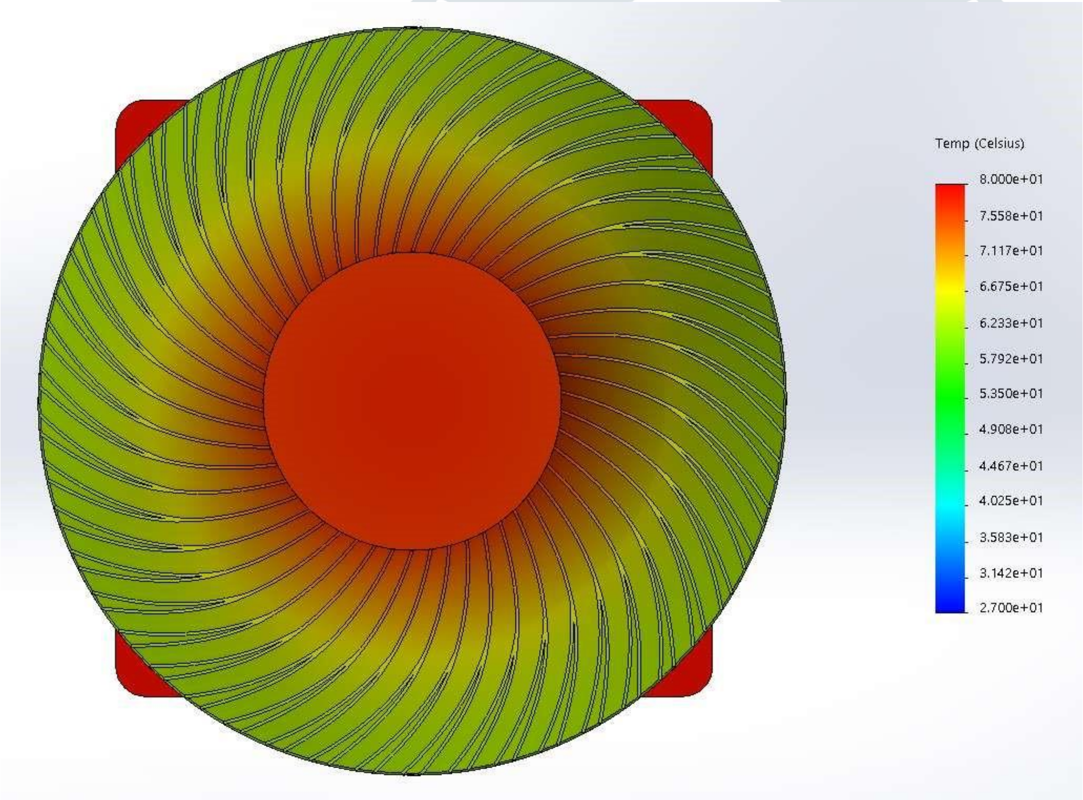

Figure 5.4
(Circular Split Twist Heat Sink Thermal Simulation Top view)

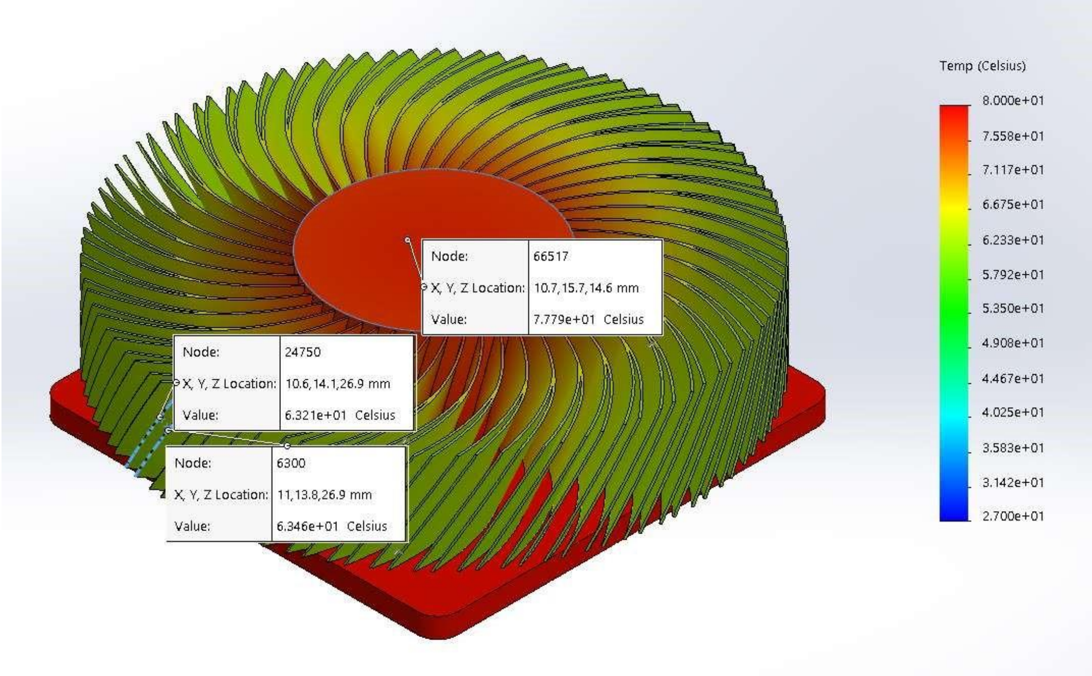

Figure 5.5
(Circular Split Twist Heat Sink Thermal Simulation Node Temperature at Fin Tip and center)

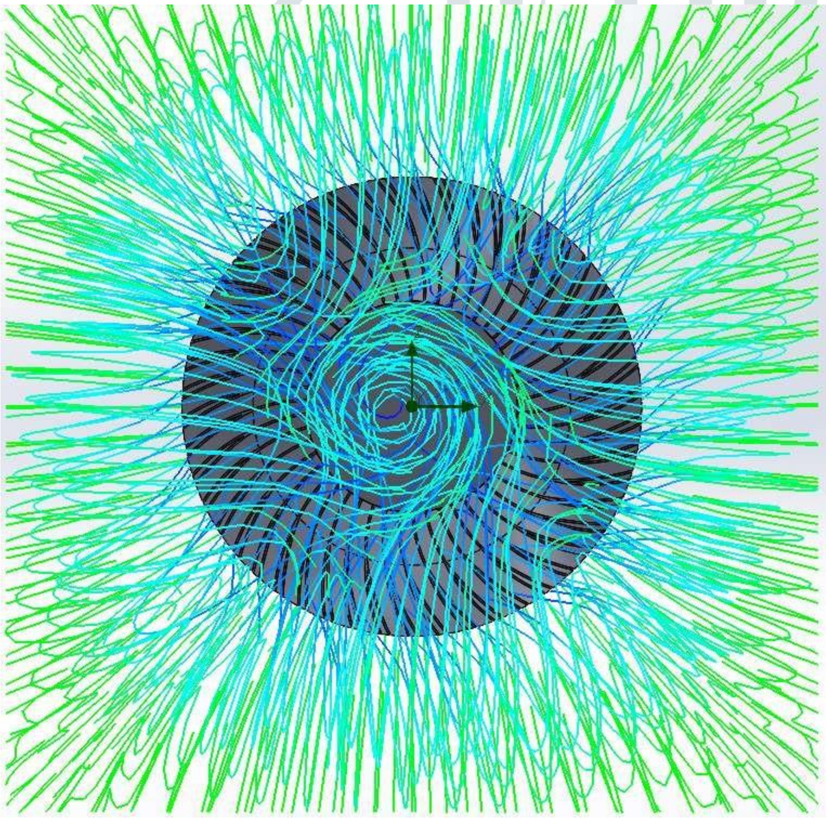

Figure 5.6
(Circular Split Twist Heat Sink Flow Simulation Axial View)

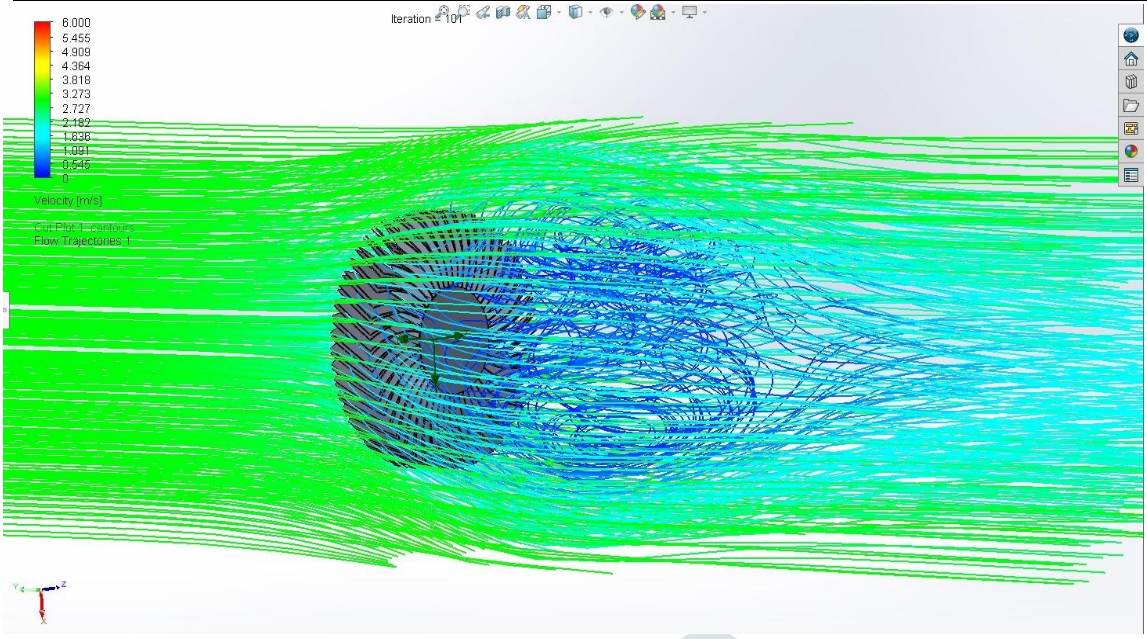

Figure 5.7
(Circular Split Twist Heat Sink Flow Simulation Lateral View)

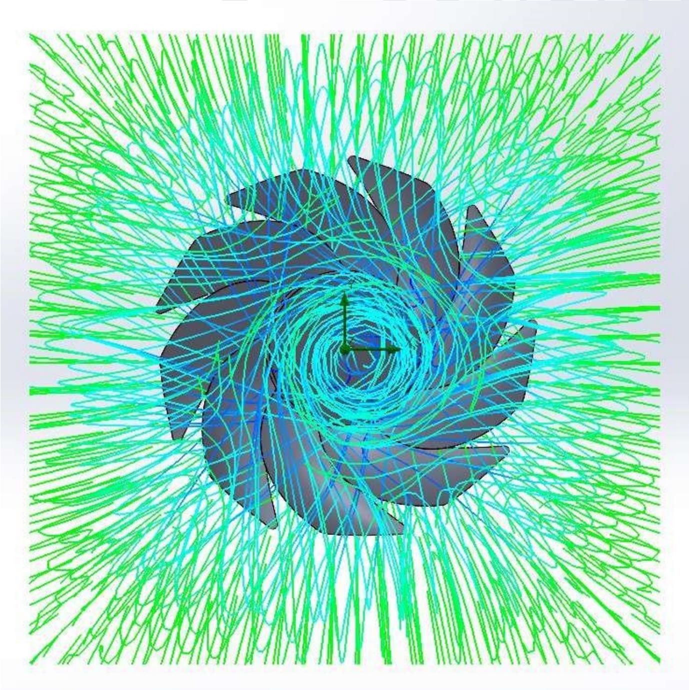

Figure 6.1
(Static Airflow Fan 9 Blades Flow Simulation Axial View)

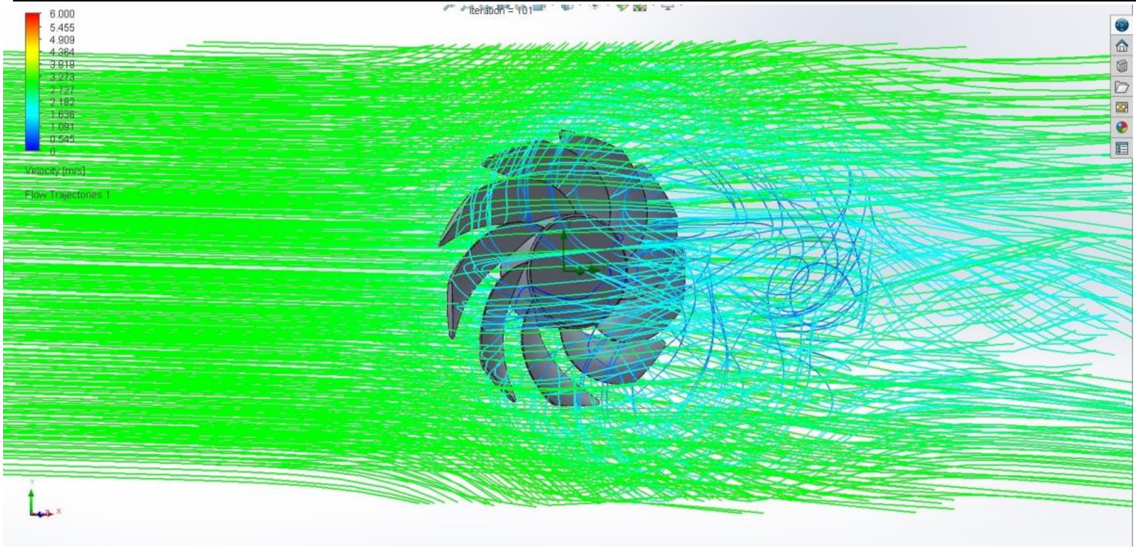

Figure 6.2
(Static Airflow Fan 9 blades Flow Simulation Lateral View)

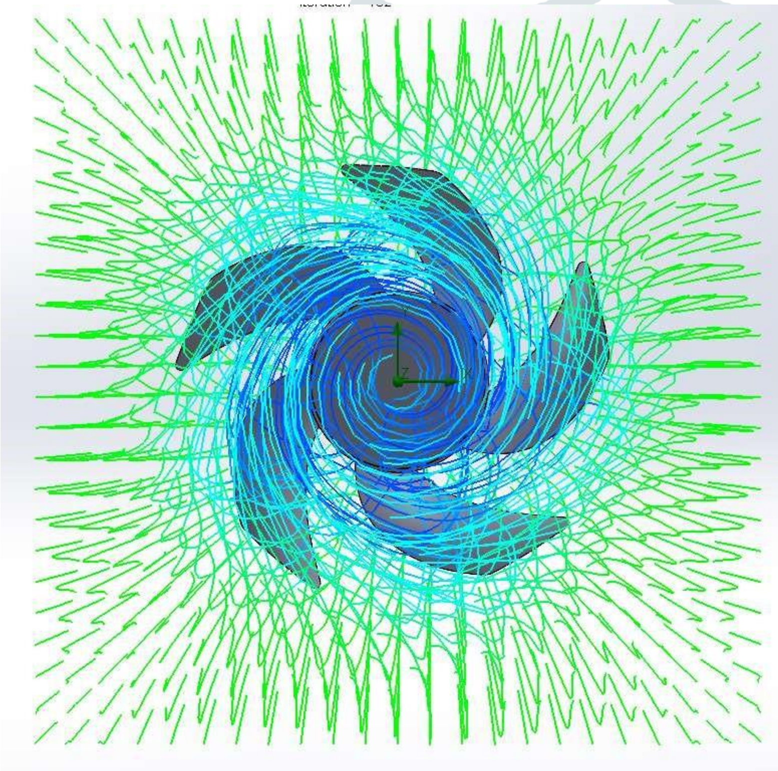

Figure 7.1
(Static Airflow Fan 5 Blades Flow Simulation Axial View)

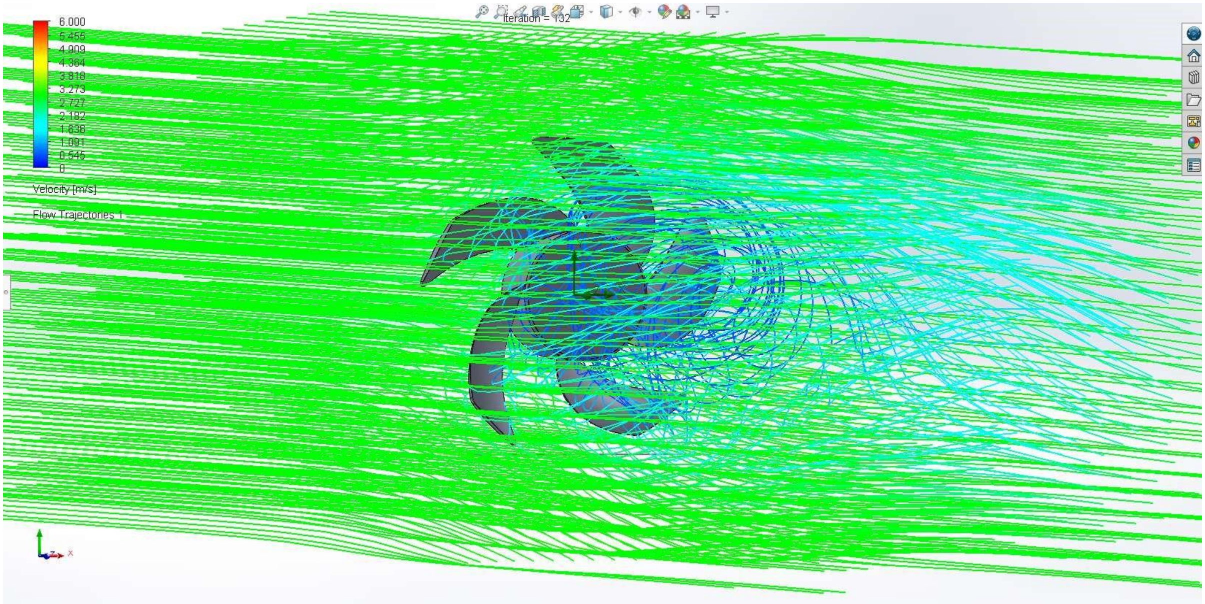

Figure 7.2
(Static Airflow Fan 5 blades Flow Simulation Lateral View)

Static Airflow Fan having fewer blades showed better air mixing results as the fan was kept static and more number of blades created a low pressure and low velocity region behind the blades, but that doesn't suggest that having more blades is less flow intensive, it just shows that more blades displaces more air out, in which case it will produce more airflow in less RPM.

More blades displaces more air in static airflow fan, which in turn generates more airflow at same RPM.

- Static Airflow Fan is made of aluminium, 10 mm diameter of solid core, it is curved and it is having 9 blades, outer diameter of blades is kept 24 mm. Another Static airflow fan is also chosen having 5 blades instead of 9.
- Circular split is chosen instead of circular straight as the area increases and the rate of heat dissipation at the tip increases drastically, which is good for our study.

6 RESULTS

Thermal Simulation was conducted by different types of fins over the Processor Chip, for our Thermal Simulation we have considered the temperature of CPU chip to be 80 degree Celsius , temperature can even go above that even 100 degrees, but for our calculation we have considered it to be 80 degree Celsius. 1060 Aluminium Alloy is used for the Heat Sink and pure Copper is used for Processor chip, and 60 W/m^2 convection coefficient is used.

Ideally we want air or fluid to twist in flow simulation, the more the twist the better will be for our analysis as it will take more time to pass through, and air mixing will be more, which will in turn increase the overall thermal heat dissipation.

- In Thermal Simulation of Circular Straight we can observe that the temperature at the tip of the fins has reached 70 degrees Celsius as shown in (Figure 2.5).
- In Flow Simulation of Circular Straight we can observe no air mixing, which means that air will pass straight through it as shown in (Figure 2.6).
- In Thermal Simulation of Circular Twist we can observe that the temperature at the tip of the fins has reached 69 degrees Celsius as shown in (Figure 3.5). We can observe that twist also increases the rate of heat dissipation up till some extent, in our case 1 degree Celsius drop of temperature.
- In Flow Simulation of Circular Straight we can observe some air mixing, which means that air will pass through with some twist it as shown in (Figure 3.6).

- In Thermal Simulation of Circular Split Straight we can observe that the temperature at the tip of the fins has reached 64 degrees Celsius as shown in (Figure 4.5). Due to the split more heat is able to dissipate, even less fins also impact better for our study.
- In Flow Simulation of Circular Split Straight we can observe better air mixing, as shown in (Figure 4.6). The mixing seems to be a little better than the Circular Twist.
- In Thermal Simulation of Circular Split Twist we can observe that the temperature at the tip of the fins has reached 63 degrees Celsius as shown in (Figure 5.5). Due to the split more heat is able to dissipate, even less fins also impact better for our study. Here also we can observe 1 degree Celsius drop in temperature due to the twist.
- In Flow Simulation of Circular Split Twist we can observe even better air mixing, as shown in (Figure 5.6). Here we have our best case scenario, best mixing is observed overall.

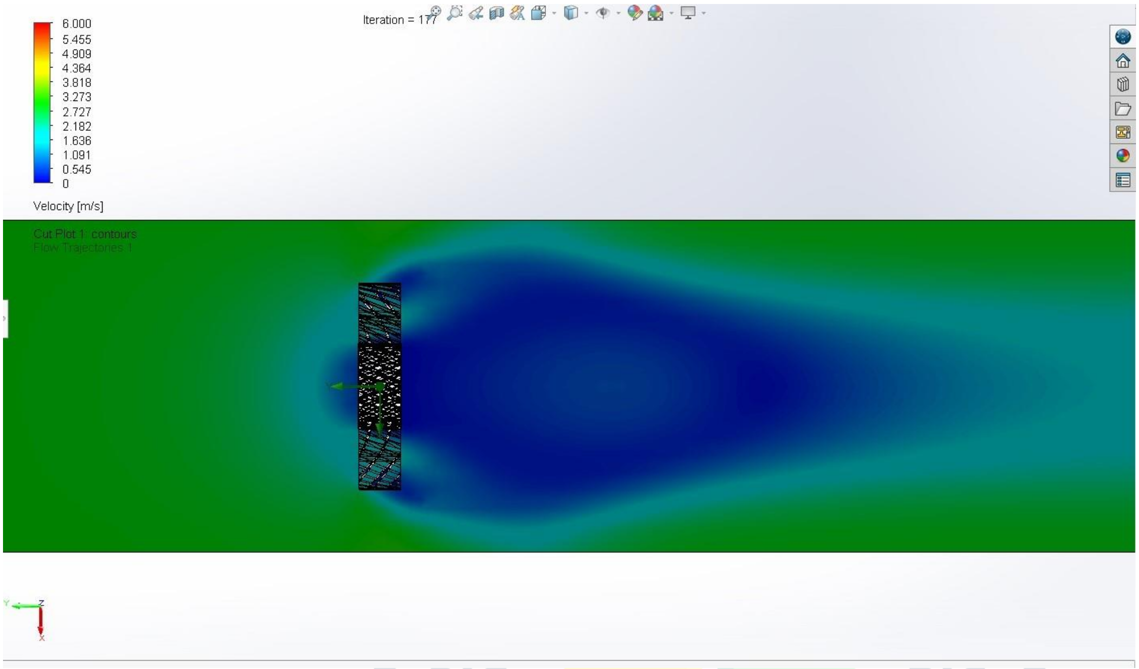

Figure 5.8
(Circular Split Twist Flow Simulation Cut Plot of Air Velocity)

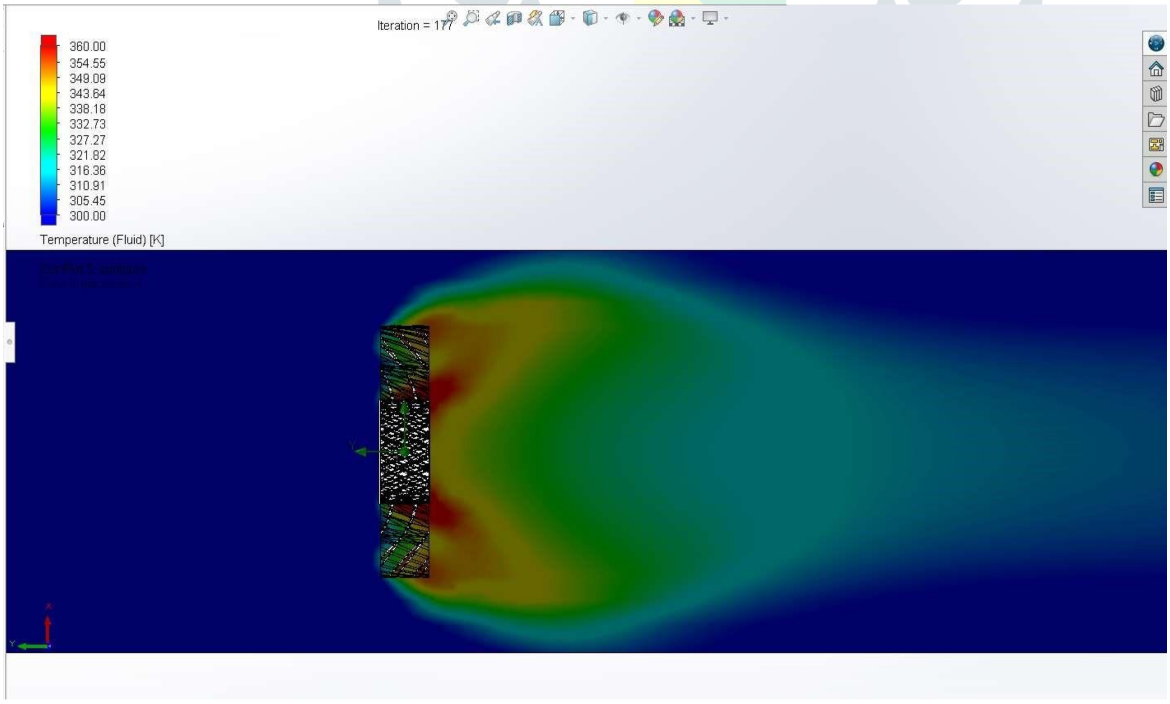

Figure 5.9
(Circular Split Twist Flow Simulation Cut Plot of Air Temperature)

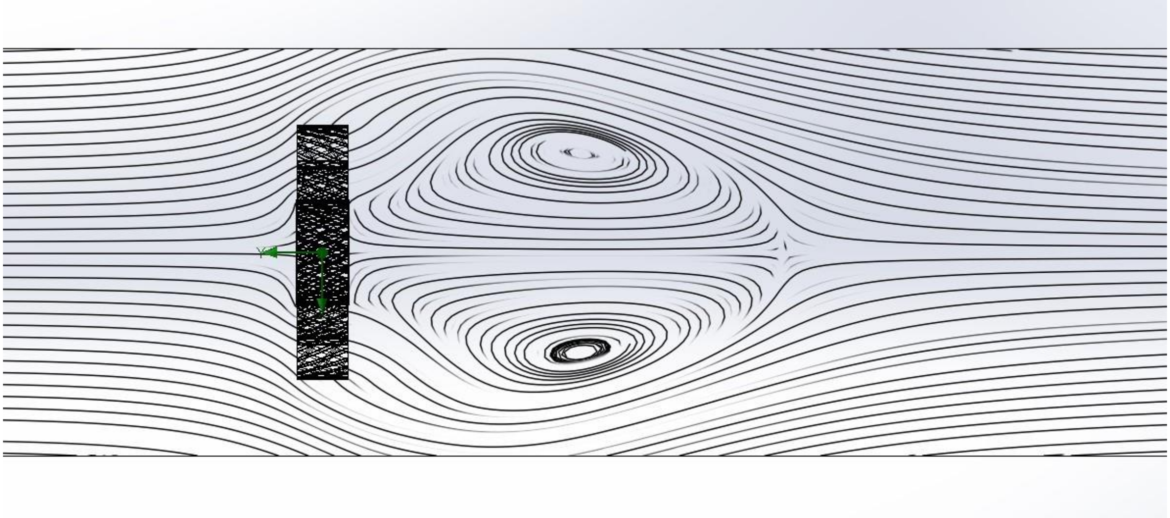

Figure 5.10
(Circular Split Twist Flow Simulation Streamlines Flow of Air)

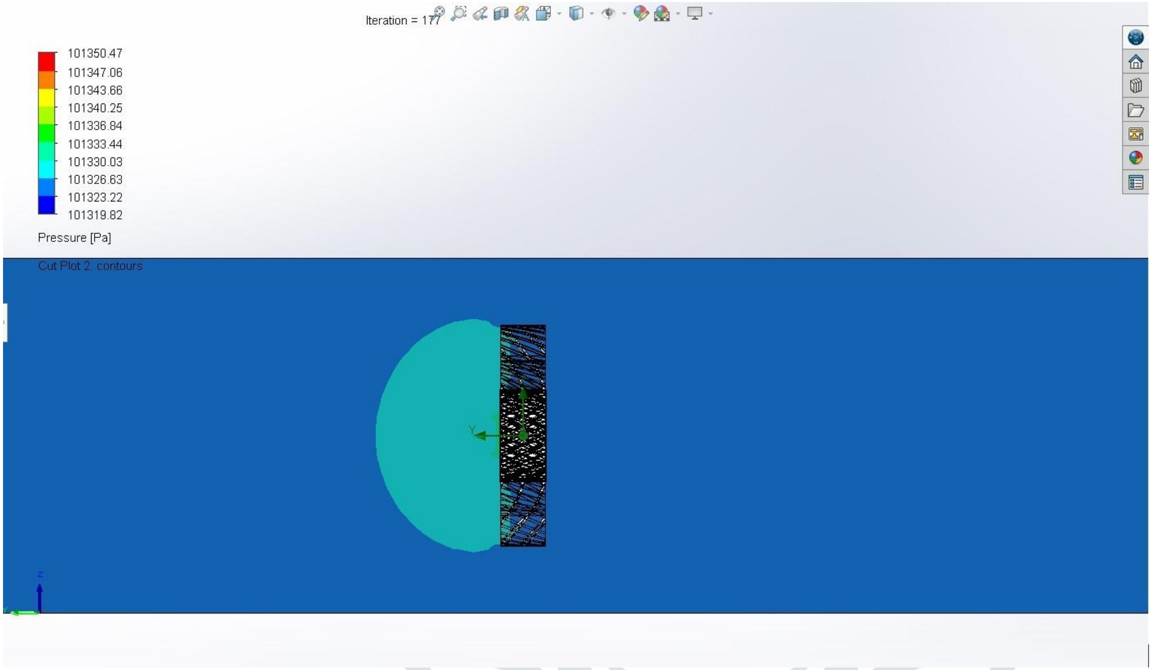

Figure 5.11
(Circular Split Twist Flow Simulation Cut Plot of Air Pressure)

- In the cut plot of Velocity (Figure 5.8) of air we can observe low velocity region that's created due to the fin,
- In the cut plot of Temperature (Figure 5.9) we can observe the temperature variations as computed.
- In the cut plot of Streamline Flow (Figure 5.10) we can observe the low velocity regions which represents the swirling of our working fluid which in this case is air.
- In the cut plot of Pressure (Figure 5.11) we can observe low pressure region that's created behind the flow of the Circular Split Twist Fin.
- In the cut plot of Temperature (Figure 5.12) we can observe temperature flow simulation of streamline trajectories.

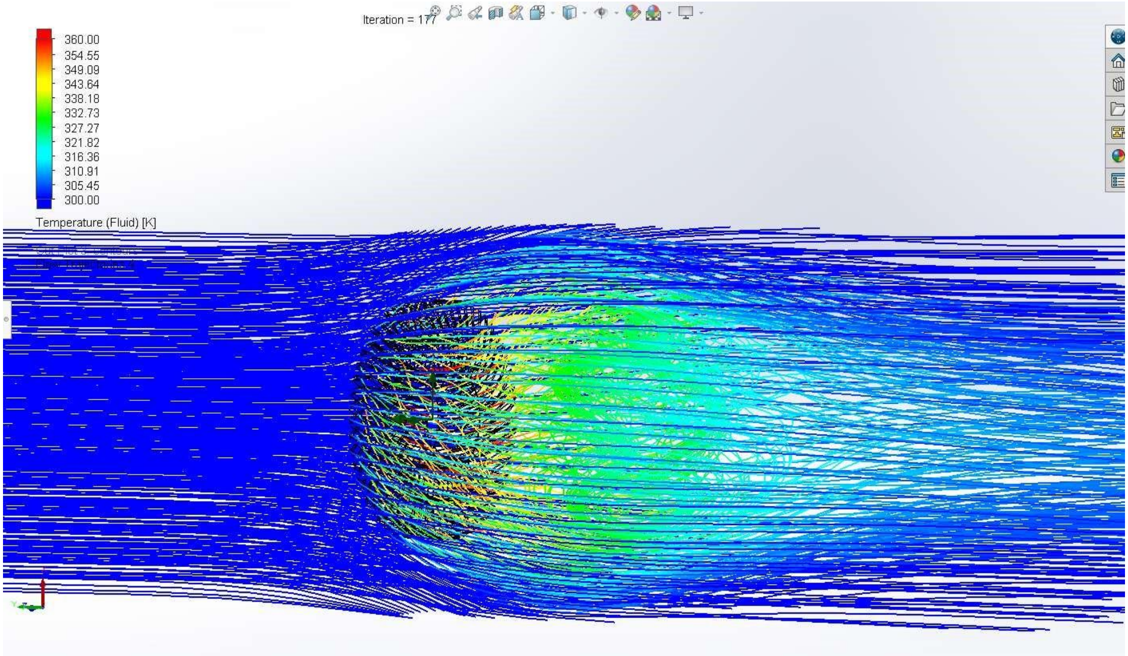

Figure 5.12
(Circular Split Twist Flow Simulation Flow Trajectories of Temperature of Air)

- In the cut plot of Temperature of Solid in (Figure 5.13 & 5.14) we can observe the high temperature of the Circular Split Twist Heat Sink, which is taken about 80 degree Celsius by implementing 30 watts heat source at one face of the Circular Split Twist Heat Sink.

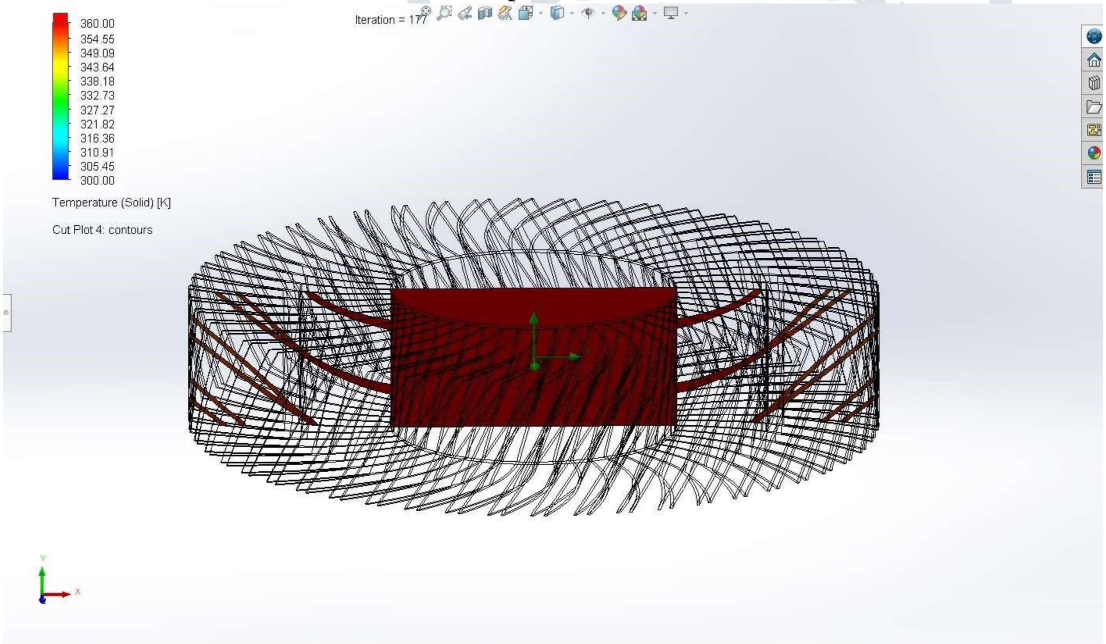

Figure 5.13
(Circular Split Twist Flow Simulation Cut Plot of Temperature of Aluminium Isometric view)

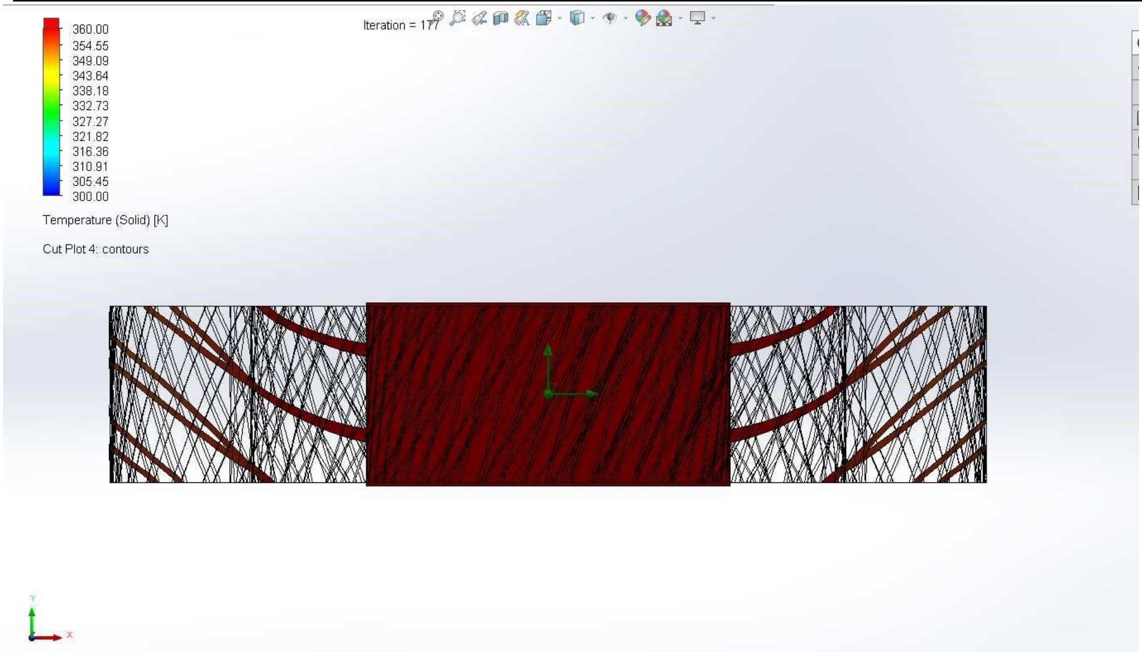

Figure 5.14
(Circular Split Twist Flow Simulation Cut Plot of Temperature of Aluminium Lateral view)

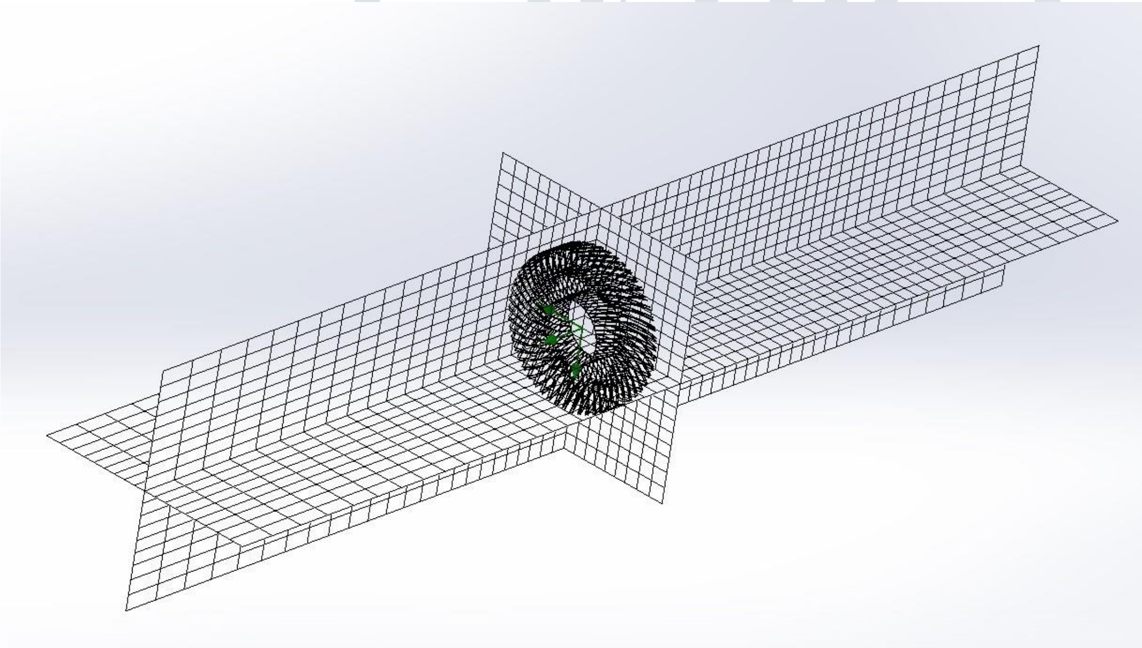

Figure 5.15
(Circular Split Twist Flow Simulation Mesh Generation)

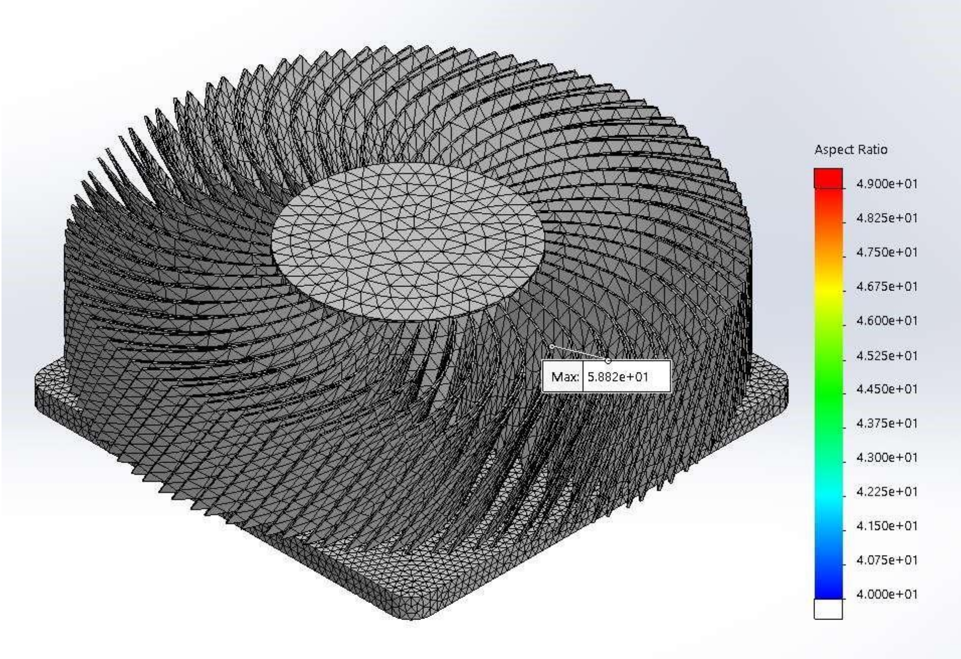

Figure 5.16
(Circular Split Twist Simulation Mesh Generation Isometric View)

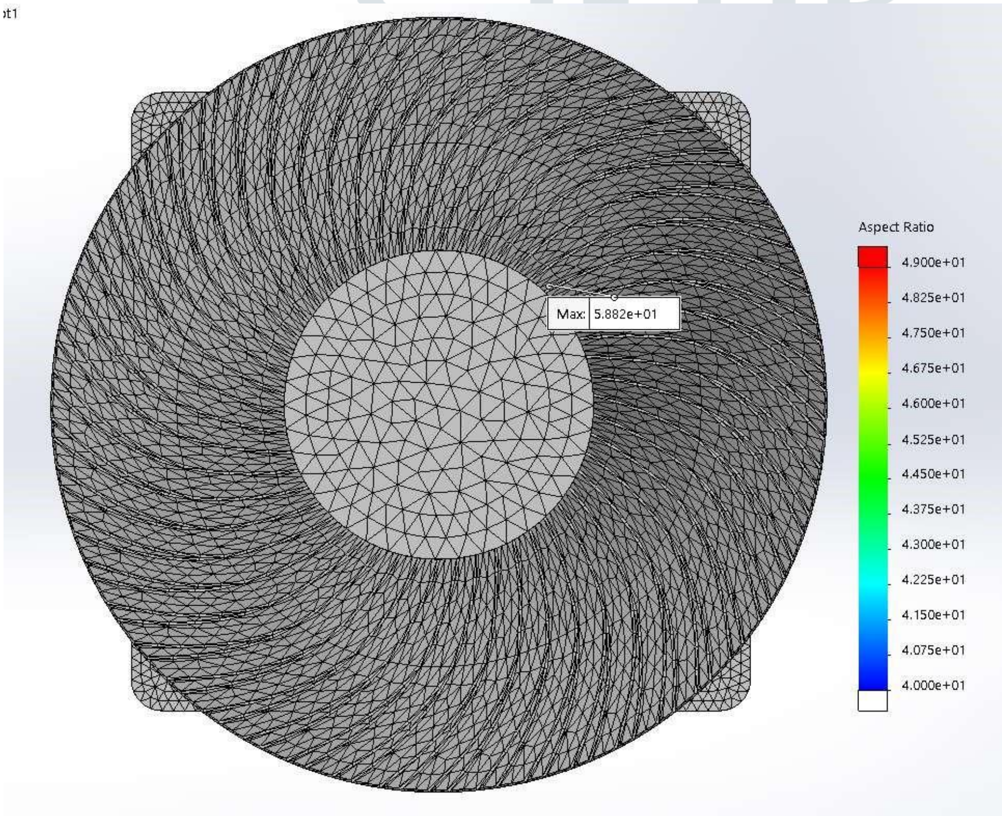

Figure 5.17
(Circular Split Twist Simulation Mesh Generation Top View) (Figure 5.15) illustrates the meshing done for our Flow Simulation Analysis, and (Figure 5.16 & 5.17) illustrates the meshing done for our Thermal Simulation.

Thermal Flow Simulation was performed inside SOLIDWORKS Flow Simulation by adding a 30 Watt heat source on the circular face of Circular Split Twist, Temperature of about 80 degree Celsius was generated by the heat source, Flow Simulation was performed, Temperature, Velocity, Pressure cut plots of our working fluid is recorded which is air in our case, and temperature cut plot of our solid was also recorded.

Simulation Performed in ANSYS 2019

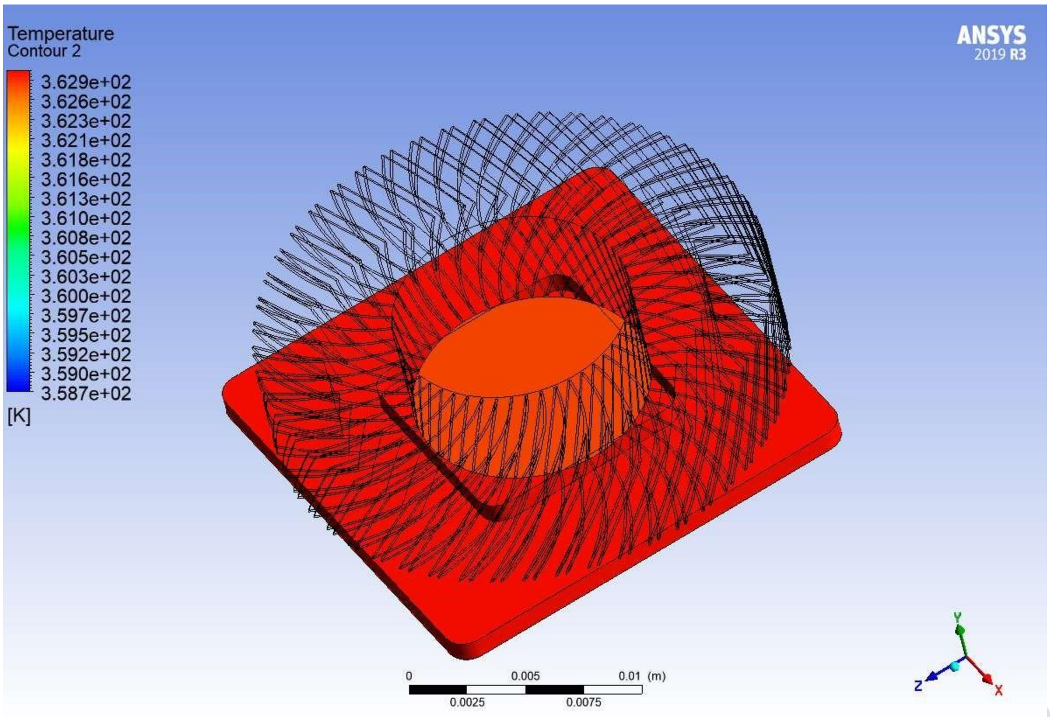

Figure 6.1
(Circular Split Twist Assembly Simulation Thermal view of Processor Chip)

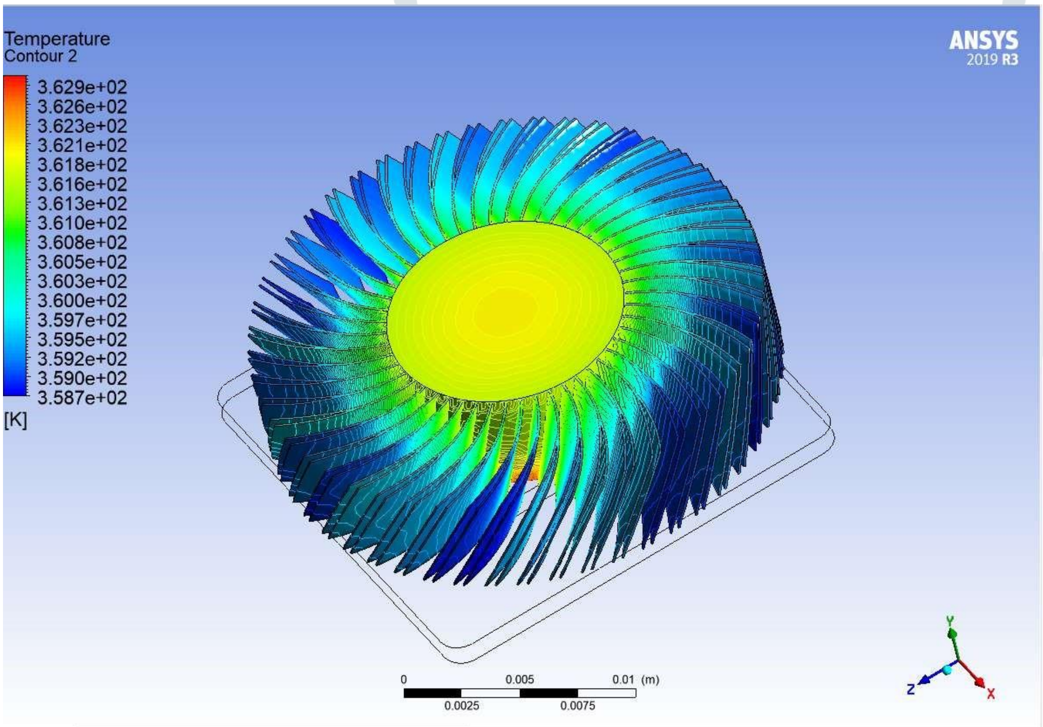

Figure 6.2
(Circular Split Twist Assembly Simulation Thermal view of Heat Sink)

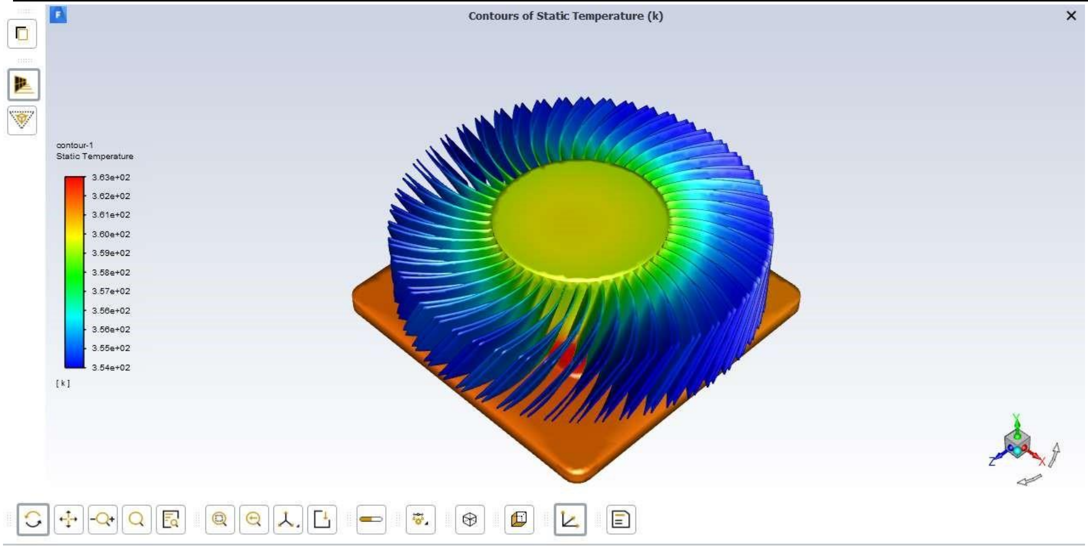

Figure 6.3
(Circular Split Twist Assembly Simulation Thermal view)

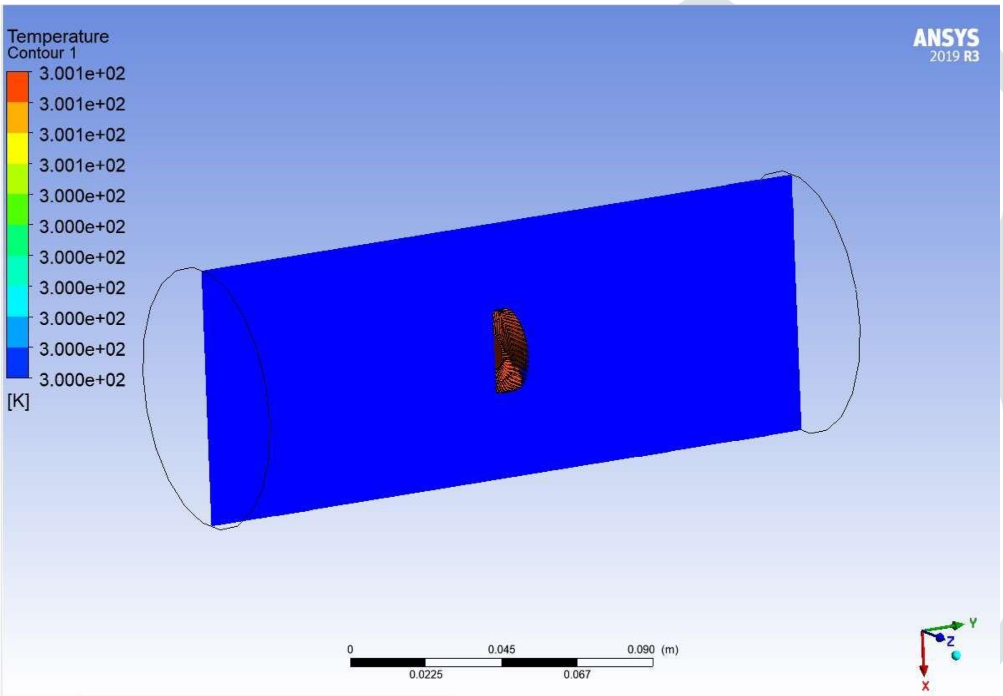

Figure 6.4
(Circular Split Twist Assembly Simulation Thermal Section view)

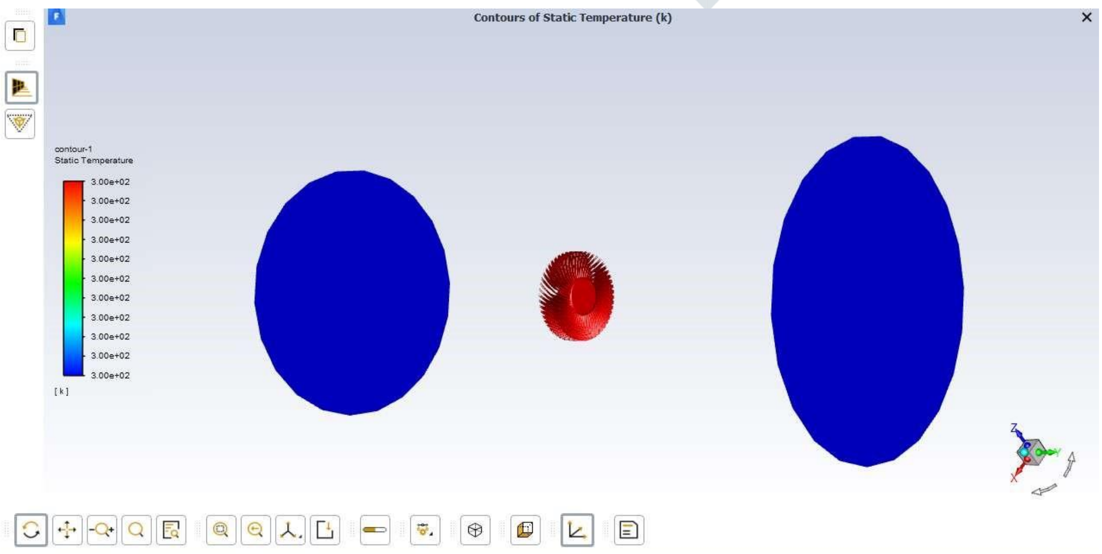

Figure 6.5
(Circular Split Twist Assembly Simulation Thermal Dynamic view)

7 CONCLUSION

In our study we can conclude that Circular Split Twist, showed overall the best air mixing which we performed in its Flow Simulation due to the splitting and twisting of the fin the overall heat dissipation was the highest as compared to other Fins as we can observe after our Thermal Simulation, as well as in our Thermal Flow Simulation by incorporating our heat source. All the study is carried out in SOLIDWORKS software.
After doing some research we came into a conclusion that the simulation performed by the current version of SOLIDWORKS software that is 2021 is not adequate to simulate and identify the thrust produced by 5 blades vs 9 blades given the model, we can however generate some rotation of our blade using “Rotating Region” function but it cannot determine the generated thrust by blades, so this research is at halt until we find anything that help solve this problem.

ANSYS Simulations were carried out in 2 phases:
In phase 1 I tried to simulate thermal simulation of the assembly of Circular Split Twist and Processor Chip, like that I did perform in SOLIDWORKS, the results were somewhat satisfactory
In phase 2 I tried to perform Thermal Flow Simulation Analysis using ANSYS fluent, where the results were not at all satisfactory as no heat dissipation zone is created like that in SOLIDWORKS.

REFERENCES

[1] Cengel, Y. A., & Cimbala, J. M. (2017). Fluid mechanics: Fundamentals and applications (4th ed.). McGraw-Hill Education.
[2] Çengel, Yunus A., and Michael A. Boles. 2001. Thermodynamics: an engineering approach. Boston: McGraw-Hill.
[3] R.Mohan and Dr.P.Govindarajan. ―Thermal analysis of CPU with composite pin fin heat sinks ‖,International Journal of Engineering Science and Technology ,Vol. 2(9), 2010, 4051-4062.
[4] H. Jonsson and B. Moshfegh ,‖ Modeling of the thermal and hydraulic performance of plate fin, strip fin, and pin fin heat sinksinfluence of flow bypass‖, IEEE Trans. Compon. Packag. Technol., 24 (2000) 117-123.
[5] Arularasan R and Velraj R, 2008. “CFD analysis in a heat sink for cooling electronic devices”, International Journal of the computer, the internet and management, Volume 6, pp.1-11.
[6] Kwang-soo Kim, Myong–hee Won, Jong-wook Kim, Byung-joon Back (2003), “Heat pipe cooling technology for desktop PC CPU,” Applied Thermal Engineering, pp 1137-1144.
[7] A. Bejan and G. A. Ledezma, “Thermodynamic optimization of cooling techniques for electronic packages,” International Journal of Heat and Mass Transfer, pp. 1213-1221, Volume 39.
[8] Dong-Kwon Kim, Sung Jin Kim, Jin-Kwon Bae 2009, Comparison of thermal performances of plate-fifin and pin-fifin heat sinks subject to an impinging flflow, International Journal of Heat and Mass Transfer 52 3510–3517.
[9] Li and Chao 2009, Measurement of performance of plate-fifin heat sinks with cross flow cooling International Journal of Heat and Mass Transfer 52 2949–2955.
[10] R. A. Wirtz, W. Chen, and R. Zhou, “Effect of flow bypass on the performance of longitudinal fin heat sinks,” ASME J. Electron. Packag., vol. 116, pp. 206–211, Sept. 1994.
[11] K. S. Lau and R. L. Mahajan, “Effects of tip clearance and fin density on the performance of heat sinks for VLSI packages,” IEEE Trans. Comp., Hybrids, Manufact. Technol., vol. 12, pp. 756–765, Dec. 1989.
[12] B. Boesmans, F. Christiaens, J. Berghmans, and E. Beyne, “Design of an optimal heat-sink geometry for forced convection air cooling of multi-chip modules,” in Proc. Eurotherm Seminar 29 “Thermal Manag. of Electron. Syst.”, Delft, The Netherlands, June 14–16, 1993.
[13] C.W. Leung, S.D. Probert, M.J. Shilston, Heat exchanger design: thermal performances of rectangular fifins protruding from vertical or horizontal rectangular bases, Appl. Energy 20 (2) (1985) 123–140.
[14] C.W. Leung, S.D. Probert, M.J. Shilston, Heat exchanger: optimal separation for vertical rectangular fifins protruding from a vertical rectangular base, Appl.
Energy 19 (2) (1985) 77–85.

[15] C.W. Leung, S.D. Probert, M.J. Shilston, Heat transfer performances of vertical rectangular fifins protruding from rectangular bases: effect of fifin length, Appl.
Energy 22 (4) (1986) 313–318.

[16] C.W. Leung, S.D. Probert, Heat-exchanger design: optimal uniform thickness of vertical rectangular fifins protruding perpendicularly outwards, at uniform separations, from a vertical rectangular 'Base', Appl. Energy 26 (2) (1987) 111– 118.

[17] C.W. Leung, S.D. Probert, Heat-exchanger design: optimal length of an array of uniformly-spaced vertical rectangular fifins protruding upwards from a horizontal base, Appl. Energy 30 (1) (1988) 29–35.

[18] C.W. Leung, S.D. Probert, Heat exchanger design: optimal thickness (under natural convective conditions) of vertical rectangular fifins protruding upwards from a horizontal rectangular base, Appl. Energy 29 (4) (1988) 299–306. [19] C.W. Leung, S.D. Probert, Thermal effectiveness of short-protrusion rectangular, heat-exchanger fifins, Appl. Energy 34 (1) (1989) 1–8.

[20] Y.M. Ko, C.W. Leung, S.D. Probert, Steady-state free-convective cooling of heat exchangers with vertical rectangular fifins: effect of fifin material, Appl. Energy 34 (3) (1989) 181–191.

[21] C.W. Leung, S.D. Probert, C.W. Rapley, Natural convection and radiation from vertically-based arrays of vertical, rectangular fifins: a numerical model, Appl.
Energy 35 (4) (1990) 253–266.

[22] A. de Lieto Vollaro, S. Grignaffifini, F. Gugliermetti, Optimum design of vertical rectangular fifin arrays, Int. J. Therm. Sci. 38 (6) (1999) 525–529.

[23] S. Baskaya, M. Sivrioglu, M. Ozek, Parametric study of natural convection heat transfer from horizontal rectangular fifin arrays, Int. J. Therm. Sci. 39 (8) (2000) 797–805.

[24] K.E. Starner, H.N. McManus, An experimental investigation of free-convection heat transfer from rectangular-fifin arrays, J. Heat Transfer (1963) 273–278. [25] C.W. Leung, S.D. Probert, Heat-exchanger performance: effect of orientation, Appl. Energy 33 (4) (1989) 235–252.

[26] A. Dayan, R. Kushnir, G. Mittelman, A. Ullmann, Laminar free convection underneath a downward facing hot fifin array, Int. J. Heat Mass Transfer 47 (12) (2004) 2849–2860.

[27] G. Mittelman, A. Dayan, K. Dado-Turjeman, A. Ullmann, Laminar free convection underneath a downward facing inclined hot fifin array, Int. J. Heat Mass Transfer 50 (13) (2007) 2582–2589.

[28] I. Tari, M. Mehrtash, Natural convection heat transfer from inclined plate-fifin heat sinks, Int. J. Heat Mass Transfer 56 (1) (2013) 574–593. [29] D. Sun, Q. Shen, L. Chen, CN Patent No. 201120167773.0, 2011 [30] H. Yüncü, G. Anbar, An experimental investigation on performance of rectangular fifins on a horizontal base in free convection heat transfer, Heat Mass Transfer 33 (1998) 507–514.