Indian Journal of Engineering & Materials Sciences
Vol. 27, October 2020, pp. 1018-1026

Numerical investigation on heat sink with fluid pockets for high power LEDs

Sangmesh[a]*, Gopalakrishna Keshava Narayana[b], Sudhakar Kumarasamy[c], Mahendran Samykano[d], Gowrishankar Thyagatur Panchaksharaiah[e] & Nagraj Patil[f]

[a]Department of Mechanical Engineering, BMS Institute of Technology and Management, Bangalore 560 064, India
[b]Jyothy Institute of Technology, Bangalore 560 082, India
[c]Faculty of Mechanical and Automotive Engineering Technology, University Malaysia Pahang, Pekan, Pahang 26600, Malaysia
[d]Department of Mechanical Engineering, College of Engineering, University Malaysia Pahang, Gambang, Pahang 26300, Malaysia
[e]Department of Mechanical Engineering, R L Jalappa Institute of Technology, Kodigehalli 561 203, India
[f]School of Engineering and Technology, Jain Deemed to be University, Bangalore 560 069, India

Received: 28 October 2019; Accepted: 31 May 2020

The present numerical study explores the use of fluid pockets in the heat sink to enhance heat transfer in high power LEDs. A robust heat sink model has been presented and evaluated the heat transfer characterization in terms of reduced LED junction temperature *via* natural convection and studied the effect of fluid flow in the pockets of heat sink. The junction temperature of the LED has been measured, for enhanced heat transfer and the results have been compared with conventional heat sink. The cooling fluid inside the fluid pockets absorbs heat generated by LEDs resulting in exchange of heat from heat sink surface to the liquid medium in the fluid pockets. The heat gain causes the fluid to flow against gravity due to density variation, raises the mixture of liquid inside the fluid pockets and flow back by gravity effect when it is condensed by the extended fin surface. The performance of the heat sink with fluid pockets has been found to be better than normal heat sink of same geometry due to its ability to conduct heat by the presence of liquid. Fluid pockets filled with de-ionized water in the heat sink have a noticeable effect on heat removal rate. A series of case studies have been done for accurate and efficient heat transfer output; these results then have been used as the benchmark to validate the experimental results. The numerical results have been found to be in good agreement with experimental results.

Keywords: ANSYS, De-ionized water, HSFP, LED, Heat sink

1 Introduction

Temperature of a LED has an adverse effect on lifetime of the LED. The luminous output of LED is entirely dependent on junction temperature by Daechan Jeon *et al.*[1]. The sophisticated packaging of LED, limited space, high temperature expected to raise the junction temperature which causes the LED to premature failure as given by Royston Marion *et al.*[2]. Further, Sung Jin Kim *et al.*[3] used different designs of heat sinks to minimize the junction temperature. S C Wong *et al.*[4] have conducted numerical analysis using finite element approach for multi fin heat sink design simulating the effects of the proposed design. The results indicated a positive effect with forced convection and was limited heat dissipation was obtained for natural convection in the investigation performed by Hui Huang Cheng *et al.*[5]. Whereas, Rasool Kalbasi *et al.*[6] studied the effect of varying fin size and application of heat pipes for further reduction of temperature also proved to be promising. Another aspect that could be meddled with for enhancing the cooling effect is the fin geometry was performed by Qie Shen *et al.*[7]. A study performed by Shangsheng Feng *et al.*[8] on analytical solution of fin geometries on different boundary layers was found that at lower values of thermo-geometric parameter the error was high and for the higher values of the same the errors seems to be negligible and the same results have been seen by Santosh Chaudhary *et al.*[9]. The use of fluid pockets in the heat sink performed better compared to conventional heat sink studied by Sangmesh B *et al.*[10].

From the literature, it is found that numerous studies have been undertaken on fin geometries, forced convection. However a study on heat sink with fluid pockets has not been reported in the literature. Thus, the present investigation involves numerical study of the heat transfer characteristics of a heat sink with fluid pockets filled with De-ionized water as the cooling medium subjected to natural convection. The

*Corresponding author
(E-mail: sangmesh.sangu293@gmail.com, sangmeshb@bmsit.in)

results obtained by numerical study are compared with the conventional heat sink without the fluid pockets.

2 Numerical analysis

In the present work, a three-dimensional computational fluid dynamics of different configurations is modeled with the help of ANSYS workbench. The computational modeling of a Novel heat sink was completed using the commercially available ANSYS FLUENT® finite volume CFD software.

In this, explored the concept of liquid cooling for enhanced heat dissipation and evaluates their performance with that obtained using conventional heat sink. For this study, an aluminium heat sink with appropriate number of fluid pockets in the heat sink base used for LED luminaries was selected in order to understand the effect of liquid cooling via natural convection.

The heat sink with fluid pockets for varying fin lengths with De-ionized water were analyzed for cooling high-power LED. The aluminium heat sinks of dimension 200 mm× 120 mm × 20 mm have aluminum rectangular fins for a length of 100 mm and 200 mm is considered for the current study. The analysis is carried out at constant heat supply of 80W for all the cases. For this, 80W cool weight Cree make LED was chosen to apply the heat to the heat sink and examined the performance of the heat sink when 80W heat is applied.

Discretising the transport equations, considering appropriate boundary conditions and providing grid-independent solutions are the major performance parameter considered in the present investigation.

The present study explores the solution in terms of fluid flow and heat transfer for natural convection for both steady and transient conditions for given fluid pockets in the heat sink. The law of conservation of mass, momentum and energy equations used in the domain is based on assuming a fluid with constant properties; the Boussinesq approximation was used for density.

3 Solution method and boundary conditions

Appropriate input data was provided to arrive at a proper and valid solution for a given problem. The simulation study is carried out for previously published experimentation, hence the test parameters were kept same to map with the experimental results. The variation in heat flux has not been considered for the simulation. The simulations were administered for both full length and half length fins with and without fluid pockets. The thickness of thermal interface material between the heat sink and LED was ignored in the simulations. The temperature locations were defined in the post processing of the simulations to maintain consistency with experimental results. The convergence criterion of the numerical simulations depends on the outright standardized residuals of the conditions. Convergence is considered as being accomplished at the point when these residuals turn out to be less.

The grid network in the regions of LED chips and little gaps between fins were refined. The no-slip condition was connected on the wall and the wall capacities were utilized at the close wall space. The thermal contact resistance along the chip, substrate and radiator were disregarded. Wall surface was coupled at the interface between fluid and solid regions to understand the heat transfer between the thermal radiator and the outside air space. The Neumann limit condition was applied on the surfaces of LED chips which were settled on MCPCB. Outside space was at open limit condition, which maintained the weight and temperature at 1.01×10^5 Pa and 27°C respectively. The effect of different orientations (tilt angle) of the LED in line with the street light was also studied.

The following assumptions were made in the present analysis:

1. The variation in heat flux was not considered for the simulation.
2. The air flow field is incompressible, laminar and steady.
3. The thickness of thermal interface material between the heat sink and LED was ignored in the simulations.
4. The no-slip condition was connected on the wall and the wall capacities were utilized at the close wall space.
5. The thermal contact resistance along the chip, substrate and radiator were disregarded.
6. Wall surface was coupled at the interface between fluid and solid regions to understand the heat transfer between the thermal radiator.
7. The various fluid properties such as viscosity, thermal conductivity, etc. were assumed to be constant.
8. However to match with the experimental results, the temperature locations were defined in the post processing of the simulations to maintain consistency with experimental results.

Governing Equations for natural convection for a single-phase flow can be written as:

Continuity Equation:

$$\nabla.(\rho v) = 0 \quad \text{... (1)}$$

Momentum Equation:

$$\rho \frac{dv}{dt} = -\nabla p + \eta \nabla^2 v + F \quad \text{... (2)}$$

where, ρ denotes density, v is velocity, F is the net force and t is time in second

The various fluid properties such as viscosity, thermal conductivity, etc. were assumed to be constant. Daechon jeong *et al.*[1] utilized a polynomial to accurately determine the heat transfer coefficient of the air. Transient method was adopted and simple algorithm was utilized as the solution method with body force weighted for pressure, first order upwind for Momentum, Turbulent Kinetic Energy and turbulent dissipation rate. Energy equations were solved using second order upwind scheme. Second order implicit transient formulation was adopted. The average standard deviation was maintained in the range of 0.47487.

4 Mesh independency study

The Novel heat sink with fluid pockets was modeled and meshed using ANSYS workbench 16 as shown in Fig. 1. The complete geometric details of the heat sink test specimen along with fluid pockets were chosen for the mesh independency study to accomplish a numerical analysis to match with the experimental conditions. The initial mesh independence study was found to be satisfactory and it was decided to use maximum mesh elements. Among various mesh types, grid with non-conformal structure comprising tetrahedral elements was adopted for discretisation of all portions of the model. The transition of the mesh was of extremely high density in the fluid pockets and also in pocket walls and moderate densities was at the outer surfaces as shown in Fig. 2. Hexahedral cells mesh structure did not suit the current test specimen because of the presence of circular structured fluid pockets. The relevance centre setting was fine. The maximum face and element size were limited to 2 mm and localised mesh refinements were performed with the help of sizing control.

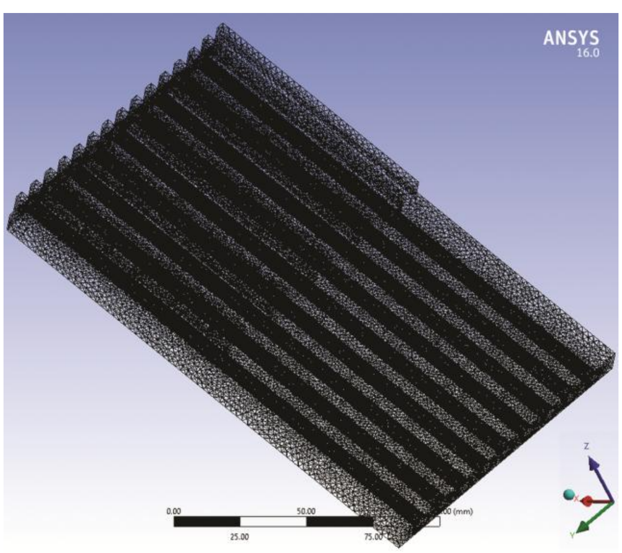

Fig. 1 — General view of the mesh on heat sink.

Body sizing was applied on the faces of the holes to accurately capture the circular geometry. The mesh in the rectangular volume of heat sink wall and circular fluid pockets were all put together. A total of 471146 nodes and elements were found to be 1260946 in all domain of the mesh geometry. A total of 242383 elements were found particularly for fluid region in the fluid pockets. Various meshed geometrical views are shown in Fig. 1 to demonstrate the mesh deployment. The mesh, in all rectangular volumes of heat sink model, was maintained by using the "quad - map" meshing arrangement scheme.

The fluid flowing regions were employed with huge elements using the cooper scheme with quad–map face.

5 Results and Discussion

In this segment, the thermal performance and airflow for different heat sink configurations, especially for a novel cooling heat sink with multiple fluid pockets at the base, are analyzed and discussed.

5.1 Effect of fluid on junction temperature

Thermal performance of the heat sink with fluid pockets was examined numerically. The numerical result demonstrates temperature contour for full length fin heat sink with fluid pockets and conventional heat sink. It was found that the heat sink with fluid pockets offered least junction temperature among all the other types of heat sink. The fluid present in the pockets was able to minimize the junction temperature due to higher heat transfer coefficient and other thermal properties. The circulation of De-ionized water in the heat sink takes place due rise in temperature, capillary and gravity effect. The temperature difference between the heat sink with fluid pockets and conventional heat sink was found to be 13°C for full length and 40°C for half-

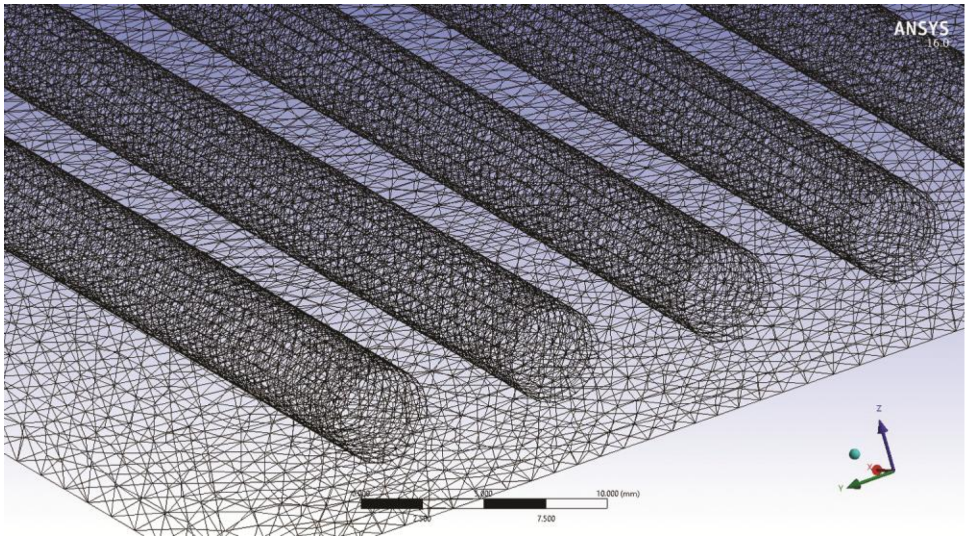

Fig. 2 — Close up view of the mesh zones shown over the heat sink along with fluid pockets.

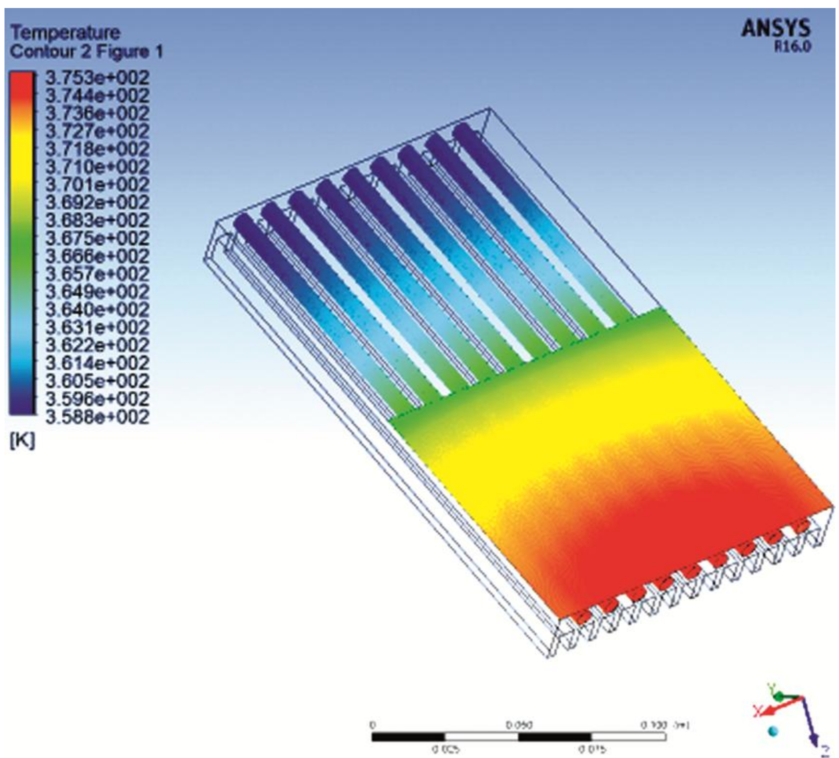

Fig. 3 — Temperature contours along with fluid pockets for full length fin heat sink with fluid pockets (HSHF01).

length fin heat sink respectively. It is noticeable from the fluid properties that water has most astounding boiling temperature, high density, latent heat, surface tension, thermal conductivity, viscosity and specific heat. This has created a viable heat exchange, prompting least junction temperature. Higher boiling temperature expands the sensible heat because of which the heat transfers rate increases. Higher thermal conductivity, surface tension and specific heat add to enhanced heat transfer.

The cooling fluid in the fluid pocket increases heat dissipation in the heat sink, accordingly enhancing cooling as shown in Fig. 3 and Fig. 4. The heat transfer enhancement because of fluid pockets in the heat sink was observed to be at the same temperature for both full length and half length fin heat sink with

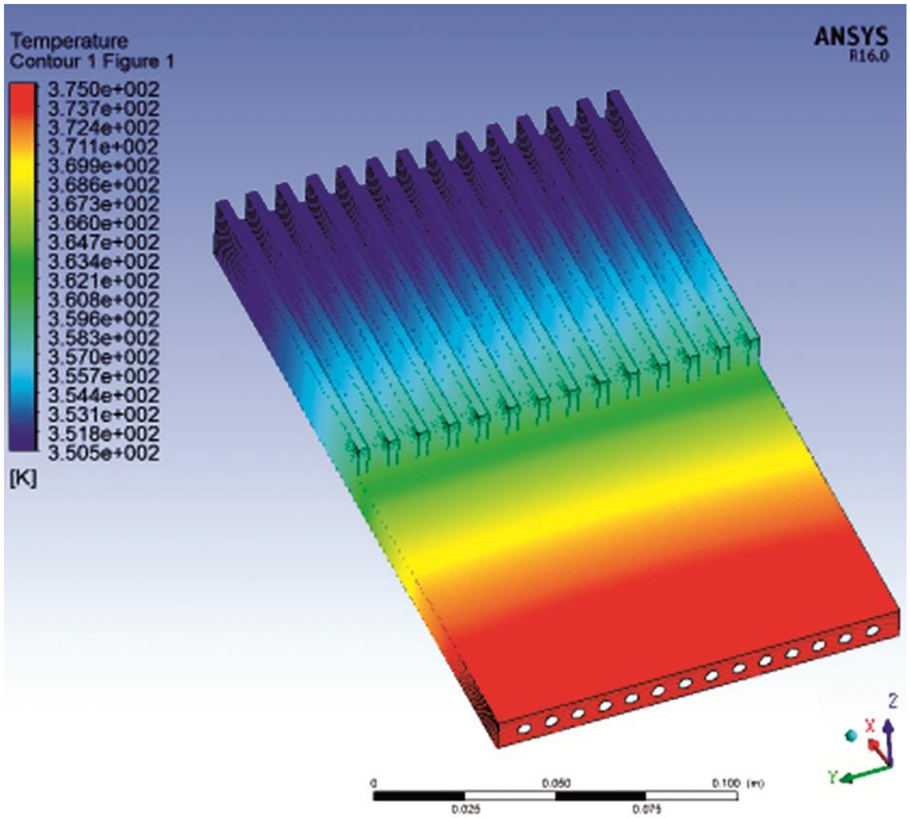

Fig. 4 —Temperature contours along with fluid pockets for half length fin heat sink with maximum fluid pockets (HSHH03).

fluid pockets. It was also demonstrated that the junction temperature was found to be lower when liquid cooling was used in the heat sink. The conventional heat sink with parallel fins proved to be the least effective.

5.2 Junction temperature in conventional heat sink

The flow pattern of atmospheric air has critical impact on the heat dissipation of LED lights. Under free convection condition, the flow pattern of atmospheric air is fundamentally dictated by the heat dissipation structure of LED illuminating presences. The heat dissipation of LEDs can be increased by enhancing the illuminators level structure to form a more sensible flow field. The heat dissipation in conventional heat sink was found to be lower than heat sink with fluid pockets due to the absence of heat transfer medium. This clearly depicts that the fluid in the heat sink rendered a positive result. In view of field synergy analysis, fluid pockets are utilized to enhance the LEDs' heat transfer structure. The homogeneous air stream leads the junction temperature of the LED is minimum and uniform over the surface of the fins. In this manner, the heat transfer execution of each type of heat sink with liquid cooling is completely utilized, and the normal junction temperature of the LEDs is reduced by 14% for full length fin heat sink as shown in Fig. 5 and approximately 40% for half-length fin heat sink as shown in Fig. 6. In comparison with the conventional heat sink, the temperature of the heat sink with fluid pockets reduced by 14 °C for full length heat sink.

5.3 Effect of fluid flow on junction temperature

It is found that the fluid flow in the pockets moves from LED junction against gravity upwards, because of the increase in thermal energy, and this flows back to the bottom due to gravitational force. This creates a circulation of the fluid within the fluid pockets resulting in an improvement in the heat transfer. From the coordination of velocity and heat flow field, as appeared in Fig. 7, the velocity and heat flow vectors are quadrature just in a little region of balances and the normal crossing point edge between them.

As per the field synergy principle, convective heat transfer is credited not exclusively to temperature contrast, fluid motion and fluid properties, but to the match level of the velocity and heat flow fields. Smaller

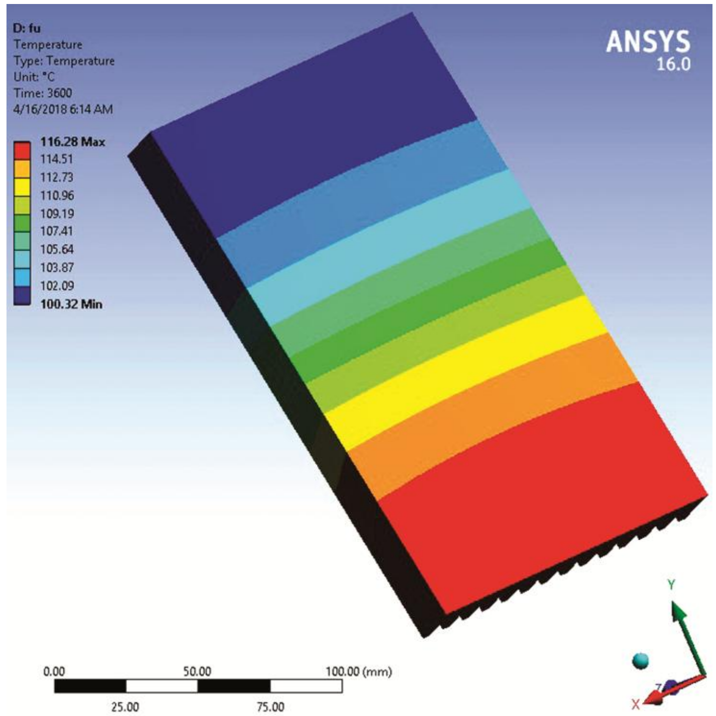

Fig. 5 — Temperature contours along the external surfaces and fins for conventional full-length fin heat sink (CHS01).

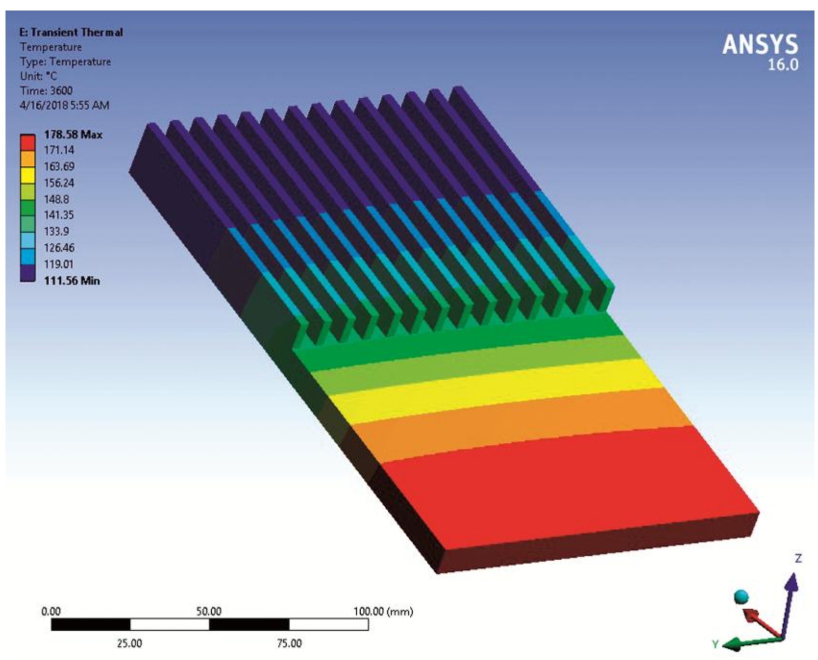

Fig. 6 — Temperature contours along the external surfaces and fins for conventional half-length fin heat sink (CHS02).

the crossing point edge amongst velocity and heat flow field, better is the heat transfer. At the point when the streamline and the isothermal line are quadrature, it implies the velocity and heat flow vector are parallel, the most satisfactory heat transfer rate is acquired. Fig. 8 and Fig. 9 show the flow appropriations of liquid cooling in the heat sink. The measurement of the diameter of the hole was 4 mm opening and 12 mm spacing between two adjoining gaps.

By contrasting HSHF01, HSHH02 and HSHH03, it is demonstrated that the fluid flow is enhanced, in the range of 2 m/s. Fig. 8 demonstrates the heat transfer

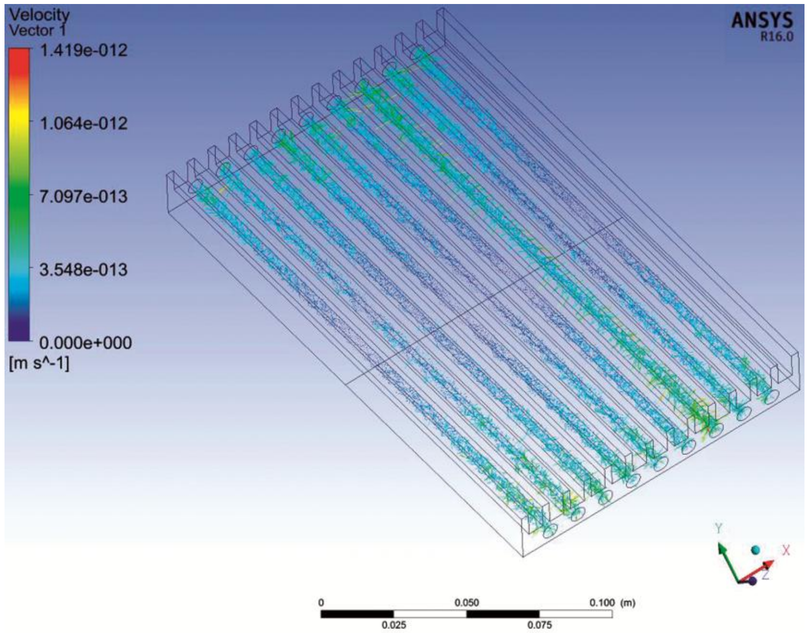

Fig. 7 — Velocity vectors along the fluid pockets for heat sink with fluid pockets for full-length fins (HSHF01).

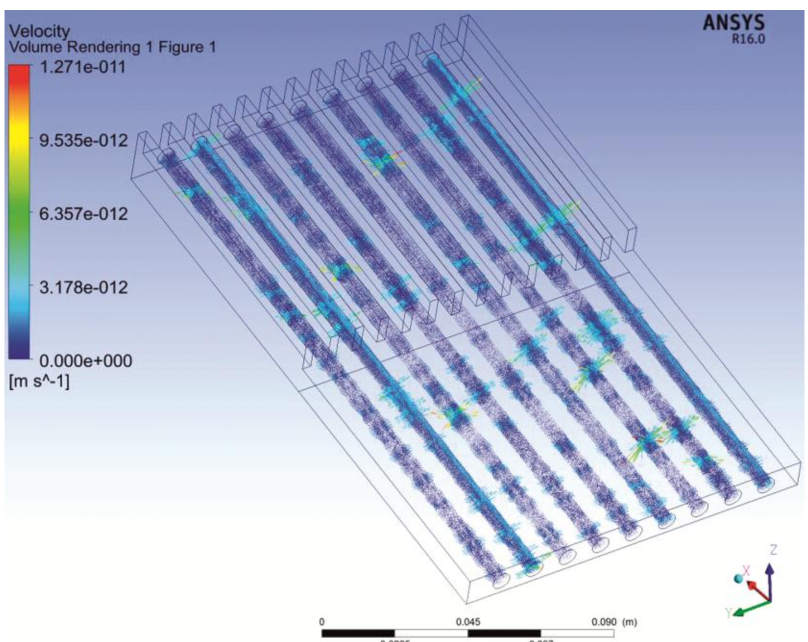

Fig. 8 — Velocity vectors along the fluid pockets for heat sink with fluid pockets for half-length fins (HSHH02).

coefficient improvement within the fluid pockets. The flow pattern surrounding the fluid pockets was seen to be improved for half-length fin heat sink in comparison to full length fin heat sink and was found to 2m/s higher than that of full-length fin heat sink.

Figure 9 demonstrates the heat transfer coefficient improvement within the fluid pockets. The flow pattern surrounding the fluid pockets was seen to be improved for half-length fin heat sink in comparison to full length fin heat sink and was found to 2 m/s higher than that of full-length fin heat sink. As indicated in Fig. 9, nine fluid pockets are disseminated along the length of heat sink in which the fluid absorbs the heat and takes it away from the LED junction.

This demonstrates that drilling through-holes in the base of the heat sink can enhance heat transfer under

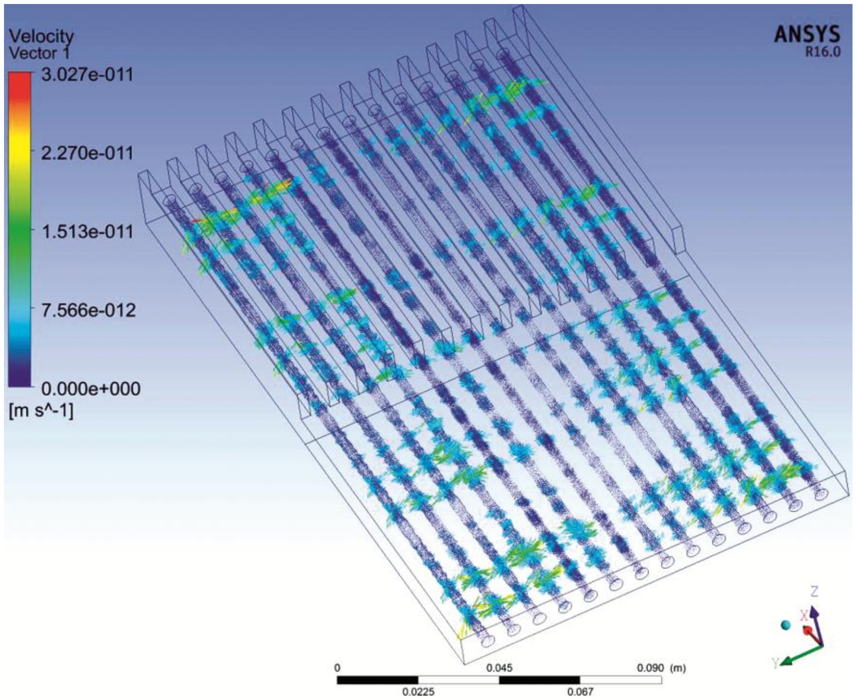

Fig. 9 — Velocity vectors along the fluid pockets for heat sink with fluid pockets for maximum number of fluid pockets (HSHH03).

Table 1 — Comparison of results.

Heat Sink Type	Designation	Junction Temperature	
		Experiment (°C)	Numerical (°C)
Conventional Heat Sink without fluid and full length fins	CHS01	110.33	116.28
Heat Sink with fluid and full length fins	HSHF01	100.23	100.3
Conventional Heat Sink without fluid and half the length fins	CHS02	116.53	178.14
Heat Sink with fluid and half the length fins	HSHH02	97.33	103.7
Heat Sink with fluid and half the length fins(Diameter of hole 4mm)	HSHH03	94.37	100.7

natural convection conditions so that heat dispersal of the LEDs is upgraded drastically. To keep the coherence of air stream field, cool air flows upwards from the bottom. After the warm air ascends to a certain height, it is cooled off due to the presence of fins and the lightness of the buoyancy effect at which point it goes down into the free space amongst various fin geometry.

Based on field synergy analysis, incorporating fluid pockets in the heat sink improve the LEDs' heat dissipation structure. It shows that drilling through holes in the heat sink can optimize the flow pattern, and then reduce the intersection angle between velocity and heat flow field under natural convection so that the heat dissipation of the LEDs is enhanced dramatically.

5.4 Comparison study

The numerical results obtained by this study are compared with the results obtained in the previously published experimental work by Sangmesh *et al.*[11]. This study involves a representative study of numerical solutions to validate the experimental results obtained earlier to ascertain the concept of liquid cooling via natural convection in a heat sink. Hence, the simulations carried out for with De-ionized water for 100% filling rate. The numerical results are in good agreement with experimental results except in one case. A series of case studies were done for accurate and efficient heat transfer output; these results were then used as the benchmark to validate the experimental results.

The present study explores the solution in terms of fluid flow and heat transfer for natural convection for both steady and transient conditions for given fluid pockets in the heat sink. The laws of conservation of mass, momentum and energy equations used in the domain are based on assuming a fluid with constant properties; the Boussinesq approximation was used for density.

The results were compared purely on output results (Junction Temperature of LED) obtained by the experiments and numerical study.

Simulation results are found to be same for HSHF01, HSHH02 and HSHH03. However, there is a negligible difference of 6°C attained for CHS01. There was a difference in case of CHS02 due to the half length fin arrangements in the heat sink. As many studies indicated that enhanced heat dissipation is observed with increase in surface area of heat sink fins but in case of CHS02 with limited in fin length would failed to extract heat where as the efficacy of the same improved with the use of fluid pockets. Hence, CHS02 obtained a maximum temperature compared to all other means of heat sinks considered in the study.

6 Conclusions

In this study, heat transfer performance of a novel heat sink with fluid pockets is presented numerically and the results were validated with experimental outcome obtained in the previous study as shown in Table 1. The thermal performance of the heat sink is evaluated in terms of reduced junction temperature of LED using De-ionized water as the heat transfer. The junction temperature reported by a heat sink with liquid is less than the conventional heat sink by at least 13%. The cooling mechanism of this device is an inspiration from conventional heat pipes where the fluid circulation is attained with condensation and evaporation of liquid. Whereas in this, the circulation of liquid may take place due to gravity force and thermal energy. The overall heat transfer of the heat sink with fluid pockets includes, both conduction and convection heat transfer. The cooling fluid inside the fluid pockets absorbs heat generated by LEDs resulting in exchange of heat from heat sink surface to the liquid medium in the fluid pockets. The heat gain causes the fluid to flow against gravity due to density variation, raises the mixture of liquid inside the fluid pockets and flow back by gravity effect when it is condensed by the extended fin surface.

References

1 Daechan J & Chan B, *Int J Heat and Mass Tra*, 113 (2017) 1086.
2 Royston M, Mendonca, Sai S Y & Chandrakant R K, *Int J Resin Eng Tech*, 41 (2015) 148.
3 Sung J K, Dong-Kwon Kim & Hwan H O, *J Heat Trans Eng*, 29 (2008) 169.
4 S C Wong & W Y Lee, *Int J The Sci*, 138 (2019) 116.
5 Hui Huang C & De Shau Haung M T L, *J Micro Ele Rel*, 52 (2012) 905.
6 Rasool K, Masoud A, Jalal A & Minh-Duc T, *J Energy*, 171 (2019) 1088.
7 Qie S, Daming S, Ya X, Tao J, Xu Z, Ning Z, Ke W & Zhiyi H, *Int J The Sci*, 28 (2015) 1.
8 Shangsheng F, Meng S, Hongbin Y, Shanyouming S, Feichen L & Tian J L, *J Appl Ther Eng*, 132 (2018) 30.
9 Chaudhary S, Kanika K M & Chaudhary S, *Ind J Eng Mater Sci*, 27 (2020) 33.
10 Sangmesh B , Gopalakrishna K, Manjunath S H, Narendra Kumar M &Vijay Kumar G, *J of Sci Ind Res*, 77 (2018) 600.
11 Sangmesh, Gopalakrishna K, Manjunath S H, Venkatesh K & Keertishekar M , *Int J Auto Mech Eng*, 14 (2017) 4846.