

International Journal of Innovative Research in Science, Engineering and Technology

(A High Impact Factor, Monthly, Peer Reviewed Journal)

Visit: www.ijirset.com

Vol. 8, Issue 3, March 2019

Performance Enhancement of Minichannel Heat sink

A.Sabari Nathan¹, R.Siluvai Silson¹, M.Syed Kani¹, G.Ugesh Pandian¹, U.Nainar²

UG Student, Department of Mechanical Engineering, Francis Xavier Engineering College,
Tirunelveli, Tamil Nadu, India ¹.

Assistant Professor, Department of Mechanical Engineering, Francis Xavier Engineering College,
Tirunelveli, Tamil Nadu, India ².

ABSTRACT: The continuing miniaturization of high-performance electronic devices has led to severely increased thermal management challenges. As per the principle of conservation of energy, all the electrical energy consumed by electronic components are evolved as heat energy. This heat energy should be dissipated to the environment to ensure the continuous functioning of electronic devices. Electronic devices have traditionally been cooled through the attachment of heat sinks. These conventional cooling methods are often incapable of providing adequate cooling for high heat flux electronic systems. New systems like heat pipes or liquid cooling systems are to be designed to manage very large heat fluxes. This work focused on the novel design of a liquid cooled heat sink targeted for the cooling of intel's brand-new Core i9 – 9980 XE processor which has the maximum allowable junction temperature of 84°C. Conventional plate fin liquid cooled mini channel heat sinks has the problem of uneven cooling due to lower temperature near Heat Transfer Fluid (HTF) inlet and higher temperature around the outlet. This problem can be overcome by the introduction of array type heat sink which has four array partitions separately fed by four fluid inlets. HTF is supplied to these inlets by a hierarchical arrangement of manifolds. The new design is numerically analysed to determine its advantages over conventional heat sink and to find the best heat transfer fluid among four kinds of fluids. HTF comparison is performed between deionized water and Al₂O₃-Water nanofluids with 1%, 2% and 3% volume concentrations. The HTF with optimal performance will be determined by calculating convective heat transfer coefficient, Nusselt number and thermal performance factor.

KEYWORDS: Minichannel Heat sink, Nanofluids, Alumina, Convective Heat Transfer

I. INTRODUCTION

Conventional minichannel heat sink design has long channels and plate fins throughout the length of the heat sink. Hence the temperature of HTF gradually increases thus the temperature difference between HTF and processor surface (ΔT) goes on decreasing. This leads to uneven surface cooling due to higher cooling rate at HTF inlet region and reduced cooling rate at HTF outlet region. It is essential to remove the heat generated in electronic devices to retain their performance. Cooling of electronic devices with minichannel heat sink is a widely used technology for dissipating heat effectively from electronic devices. Heat dissipation mainly depends on the heat sink material, minichannel design parameters and the heat transfer fluid (HTF). In this project minichannel array heat sink with hierarchical manifold configuration is designed and analyzed.

International Journal of Innovative Research in Science, Engineering and Technology

(A High Impact Factor, Monthly, Peer Reviewed Journal)

Visit: www.ijirset.com

Vol. 8, Issue 3, March 2019

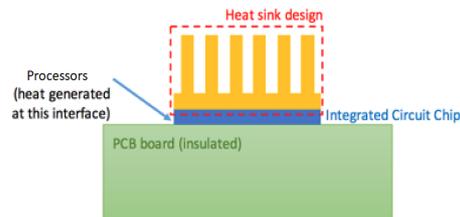


Fig.1 Heat sink

After the emergence of nanotechnology, it becomes essential to determine the performance of heat sinks using Nanofluids as heat transfer fluids. Numerical analysis has more advantages than experimental procedures as it is less time consuming and less expensive. Thermal performance of heat sinks can be well predicted using numerical analysis. This project is concerned with the numerical determination of influence of different volume concentrations of Al_2O_3 /Water nanofluid in the thermal performance of minichannel array heat sinks. In this work, Al_2O_3 /Water nanofluids with 1%, 2%, 3% volume concentrations are used as HTFs to find the suitable concentration for optimum performance.

II. LITERATURE REVIEW

Yimin Xuan “Conceptions for heat transfer correlation of nanofluids” IJHMT,2000

The nanoparticles enhance heat transfer rate by increasing the thermal conductivity of the nanofluid and incurring thermal dispersion in the flow, which is an innovative way of augmenting heat transfer process. As a new type of heat transfer medium, the nanofluid behaves more like a single-phase fluid because the discontinuous phase consists of ultrafine particles, so that the heat transfer correlation for the single-phase fluid, rather than those for the conventional solid-liquid two-phase flow, has been taken as the start point. By starting from the existing theory and correlations for the heat transfer process of pure fluids, this paper has proposed two approaches of deriving heat transfer correlation for the nanofluid.

S. Suresh, “Effect of Al_2O_3 -Cu/water hybrid nanofluid in heat transfer” ELSEVIER,2012

Heat transfer performance in a straight circular tube is amplified by suspension of Al_2O_3 -Cu hybrid nanoparticles in comparison with that of pure water. The average increase in Nusselt number for Al_2O_3 -Cu/water hybrid nanofluid is 10.94% when compared to pure water. The convective heat transfer coefficient increases with increasing Reynolds number. The experimental results of hybrid nanofluid for laminar flow showed a maximum enhancement of 13.56% in Nusselt number at a Reynolds number of 1730 when compared to pure water. The experimental results showed that dilute Al_2O_3 -Cu/ water hybrid nanofluid with 0.1% volume fraction show slightly higher friction factor when compared to pure water. The average increase in friction factor of 0.1% Al_2O_3 -Cu/ water hybrid nanofluids is 16.97% when compared to water. This reveals that dilute Al_2O_3 -Cu/water hybrid nanofluids will cause extra penalty in pumping power when compared to Al_2O_3 /water nanofluid.

Y.Raja Shekar “Heat Transfer Enhancement with Al_2O_3 Nanofluids and Twisted Tapes in a Pipe for Solar Thermal Applications”, ELSEVIER,2013

Heat transfer experiments were conducted in a pipe under low Reynolds number range using water and water based nanofluids. Heat transfer coefficient and friction factor for nanofluid in the flow path enhanced compared to water. The experimental data is compared with the data of literature and are found to be in good agreement. The increase in heat transfer coefficient in plain tube with use of nanofluids is greater by 8-12% compared to the flow of water in a plain

International Journal of Innovative Research in Science, Engineering and Technology

(A High Impact Factor, Monthly, Peer Reviewed Journal)

Visit: www.ijirset.com

Vol. 8, Issue 3, March 2019

tube. The nanofluid of 0.5% particle concentration is having highest friction factor compared to water. The Nusselt number and friction factor increases with increase of particle concentration. But, friction factor decreases with increase of Reynolds number of flow where as the Nusselt number increases. Using nanofluid with a high heat exchange can help in reduce the size of the heat exchanger or without increasing the size of the heat exchanger efficiency of the system can be improved. Further, using twisted tapes and nanofluids in the pipe flows is advantageous since it is visible from the results that the energy gained with heat exchange is more than the energy spent on pumping power.

K.P. Drummond “A Hierarchical manifold microchannel heatsink array for high-heat-flux two-phase cooling of electronics”, CTRC research publications,2018

The single-phase heat transfer coefficient was found to increase with increasing channel mass flux, which was attributed to impingement and developing flow effects. In the two-phase regime, heat transfer coefficient strongly depends on exit quality and weakly depends on channel depth and mass flux. For all channel depths and mass fluxes, heat transfer coefficient increases with increasing exit quality until a maximum is reached; after this point, the heat transfer coefficient decreases with exit quality until critical heat flux is reached. These trends match the general trends experienced in traditional microchannel heat sinks.

III. OBJECTIVE OF PROPOSED WORK

1. To design and numerically analyze a liquid cooled minichannel array heat sink, which has more even cooling throughout the surface for Intel’s Core i9 – 9980 XE processor.
2. To introduce a novel design of hierarchical manifold arrangement to efficiently supply heat transfer fluid to the array inlets.
3. To numerically determine the HTF with optimal thermal performance by calculating convective heat transfer coefficient, Nusselt number and thermal performance factor.

IV. MODELING AND ANALYSIS

DESIGN

Three-dimensional model of conventional and array heat sinks are created in Solidworks 2018 software (as shown in Figure) and the model is exported as STEP files. The STEP files were then imported into ICEM CFD, for cleanup and meshing.

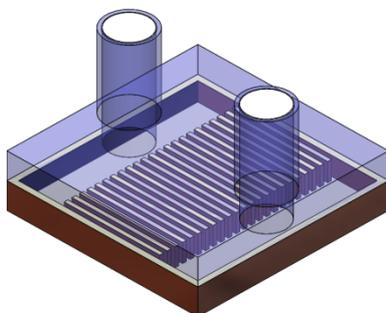


Fig.2 Conventional minichannel (plate-fin) heat sink

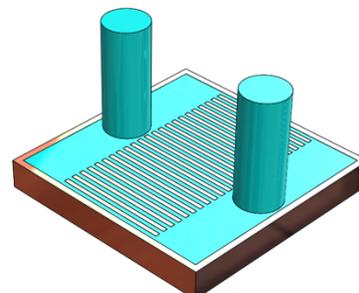


Fig.3. Extracted fluid zone of conventional heat sink

Numerical prediction of thermal performance of heat sink involves only the fluid region. Interior of the solid region and solid outside surfaces do not influence the surface flow patterns. Therefore, solid interior is removed to extract the fluid region from the assembly. The following figures show the heat sink and extracted fluid region.

International Journal of Innovative Research in Science, Engineering and Technology

(A High Impact Factor, Monthly, Peer Reviewed Journal)

Visit: www.ijirset.com

Vol. 8, Issue 3, March 2019

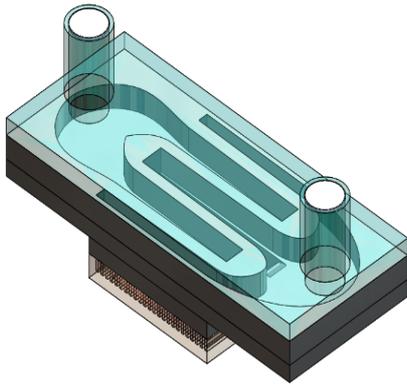


Fig.3 Array minichannel heat sink assembly

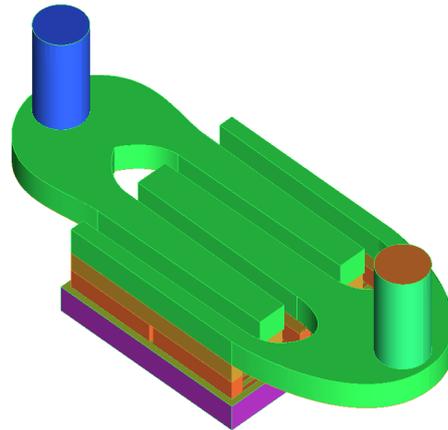


Fig.4 Extracted fluid zone of array heat sink

Manifolds in array heat sink

Array heat sink has 2×2 configuration. Each array has minichannels with 1mm width and plate fins with 1 mm thickness. Each array has rectangular inlet at the middle and two rectangular outlets at each side. This is shown in the figure of manifold 1. Manifold 2 is placed above which has lesser number of inlets and outlet ports because of the connected port arrangement. Manifold 3 has parabola and spline shaped pathways for collecting, guiding and distributing the liquid from inlet to outlet. Manifold 4 has inlet and outlet pipes. Manifolds are made up of acrylic material during experimental analysis.

The exploded view of heat sink and manifolds assembly is given in the following figure.

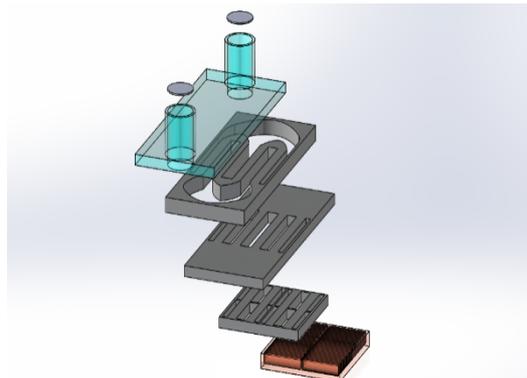


Fig.5 Exploded view of heat sink assembly

MESH

ICEM CFD is one of the best model clean up and mesh generation tool. After importing the model into ICEM CFD, topology is checked-up to find any geometrical mismatches in the CAD model. Then the fluid volume is split into a large number of small volumes called mesh elements. The total volume is surrounded by the surfaces namely inlet, pipe 1, pipe 2, walls and outlet. The inlet and outlet ducts were intentionally closed to provide boundary conditions for simulations. The flow was assumed fully developed when leaving the inlet and outlet ducts.

Meshing of fluid volume includes,

1. Surface Meshing
2. Volume Meshing

International Journal of Innovative Research in Science, Engineering and Technology

(A High Impact Factor, Monthly, Peer Reviewed Journal)

Visit: www.ijirset.com

Vol. 8, Issue 3, March 2019

Surface Mesh

The outer surface of the fluid flow path is discretized to form the surface mesh shown in figure. Surface mesh is the base for volume mesh. A total of 5.7 million triangular elements are used for creating surface mesh with about 0.8 skewness. Skewness is the degree of deviation of created element from the standard element. Equilateral triangle is considered as the standard element. Skewness value enables us to understand the quality of mesh. For a perfect mesh, skewness value is 0. For a bad mesh, the value is 1.

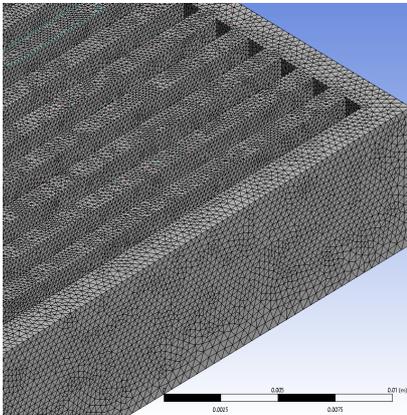


Fig.6 Surface mesh of heat sink assembly

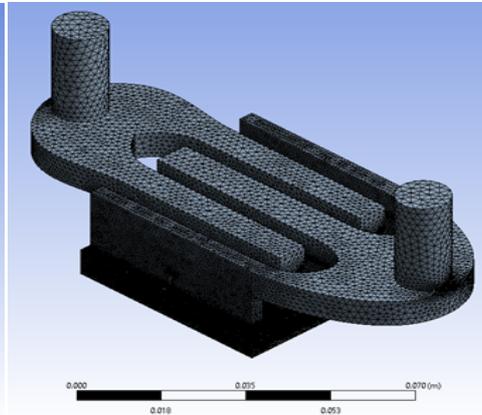


Fig.7 Surface mesh of heat sink

Volume Mesh

Quality of mesh is represented by a factor called Skewness. It is the degree of deviation of the meshed element from the standard element. Skewness value for surface mesh is 0.8, and for volumetric mesh is 0.6. The meshing method used for creating volumetric mesh is Advancing front Algorithm. This algorithm gives a smooth mesh growth rate of 1.1. Complex sections of heat sink surface area are discretized using tetrahedron elements.

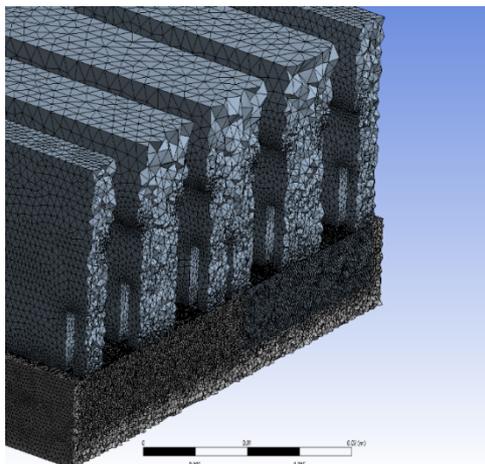


Fig.8 Cross sectional view of volume mesh

After meshing the total number of tetrahedral elements generated in the flow region is about 15.9 million. Total number of nodes are 2.7 million.

International Journal of Innovative Research in Science, Engineering and Technology

(A High Impact Factor, Monthly, Peer Reviewed Journal)

Visit: www.ijirset.com

Vol. 8, Issue 3, March 2019

V. ANALYSIS

PRE-PROCESSING (CFD ANALYSIS – PHASE I)

Material Properties

Properties of Heat Transfer Fluids

The HTFs selected for the analysis are,

1. Water
2. Al₂O₃- Water Nanofluid (1% vol. concentration)
3. Al₂O₃- Water Nanofluid (2% vol. concentration)
4. Al₂O₃- Water Nanofluid (3% vol. concentration)

Alumina (Al₂O₃) is the most common nanoparticle used by many researchers in their experimental works. Many efforts have been made to study the thermal conductivity of nanofluids. The summary of experimental studies on the thermal conductivity of Al₂O₃-based nanofluids. In this project Al₂O₃-Water Nanofluid with 1%, 2% and 3% volume concentration are selected for investigation. Specific gravity and Thermal conductivity are assumed as constants. These values are obtained from previous studies and tabulated below.

Material	Specific heat (J/kg K)	Density (kg/m ³)	Thermal conductivity (W/mK)	Viscosity (mPa.s)
Water	4180	1000	0.576	1.00
Al ₂ O ₃ - H ₂ O (1% vol.)	4060.8	1023.7	0.620	1.12
Al ₂ O ₃ - H ₂ O (2% vol.)	3947.6	1050.8	0.634	1.28
Al ₂ O ₃ - H ₂ O (3% vol.)	3839.9	1077.8	0.655	1.57

Boundary Conditions

A heat flux of 103125 W/m² is applied to the base of heat sink. Heat flux to be applied to the bottom of the heat sink base is calculated as follows.

Heat Spreader (Lid) Size 40mm×40mm

$$\text{Lid Area} = 1600\text{mm}^2 = 16\text{cm}^2 = 0.0016 \text{ m}^2$$

Maximum heat transferred through the Lid (at full load), Q = 165W

$$\text{Heat flux} = \text{heat transferred} / \text{Lid Area} = 165 / 16 = 10.3125 \text{ W/cm}^2$$

International Journal of Innovative Research in Science, Engineering and Technology

(A High Impact Factor, Monthly, Peer Reviewed Journal)

Visit: www.ijirset.com

Vol. 8, Issue 3, March 2019

Heat flux = 103125 W/m²

Wall boundary condition is assigned for all the other surfaces which do not participate in heat transfer or which do not allow fluid through them. These types of walls are also known as adiabatic walls

Sl.No	Region	Boundary Condition	Value
1	Inlet	Mass flow	0.001 kg/s
2	Inlet	Temperature	300 K
3	Outlet	Pressure	101325 Pa
4	Heat Sink Base	Heat flux	103125 W/m ²
5	Other surfaces	Adiabatic wall (Convective heat transfer coefficient 'h' is zero)	h = 0 W/m ² K

SOLUTION PHASE (CFD ANALYSIS – PHASE II)

The solution phase includes solution scheme selection, discretization method selection, solution initialization, residual control etc. Implicit numerical algorithm is used as the formulation for solving this problem in ANSYS Fluent. In addition, the parameters namely flow, turbulent kinetic energy-*k* and turbulent dissipation rate- ϵ are discretized by second-order upwind scheme.

The computing times for a flow simulation depend on the choice of numerical algorithms and data structures and linear algebra tools, stopping criteria for iterative solvers, discretization parameters (mesh quality, mesh size, time step), time step and convergence rates for outer iterations. Programming language (most CFD codes are written in Fortran) many other things (hardware, vectorization, parallelization etc.). The quality of simulation results depends on the mathematical model and underlying assumptions.

In this work, Solution is initialized with standard initialization from inlet zone. Gauge pressures and directional velocities are considered as zero initially. Stopping criteria is set by choosing maximum value of residuals as 10e-6.

V. RESULTS AND DISCUSSION

This project compares the thermal performance of array heat sink with conventional plate fin heat sink. The heat transfer fluids used for this analysis are water and Al₂O₃-water nanofluid with 1%, 2% and 3% volume concentrations. Thermal performance is determined by means of convective heat transfer coefficient (*h*), Average Nusselt number (*Nu*) and thermal performance factor (*T_f*). Convective heat transfer coefficient and average Nusselt number are directly obtained from the fluent results.

The important goal of this work is to maintain the temperature of the processor heat spreader below 84°C (< 357K). Minimum, maximum and average values of temperature at any surface can be found directly from the fluent results. In this case the surface named as heat flux wall is equivalent to the heat spreader surface. Thus minimum, maximum and

International Journal of Innovative Research in Science, Engineering and Technology

(A High Impact Factor, Monthly, Peer Reviewed Journal)

Visit: www.ijirset.com

Vol. 8, Issue 3, March 2019

average values of temperature at heat flux wall were found and it is verified that there is no temperature rise above the threshold temperature level at any point of the surface. The above table compares the temperature levels of all the analyzed models

Temperature contour of heat flux wall

Conventional plate-fin heat sink

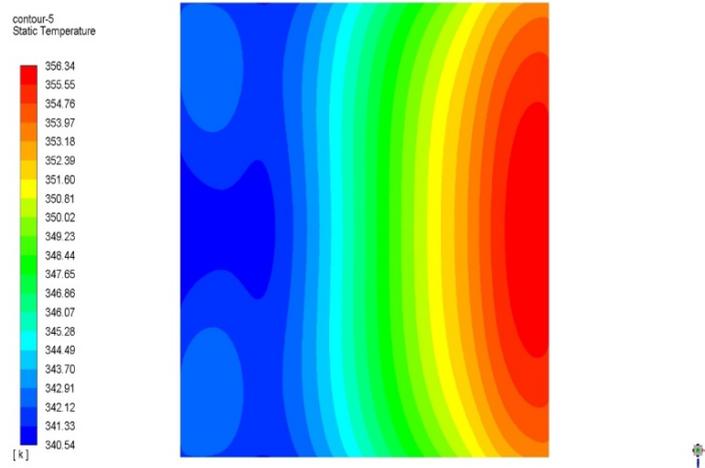


Fig.33. Temperature contour of heat flux wall (Conventional)

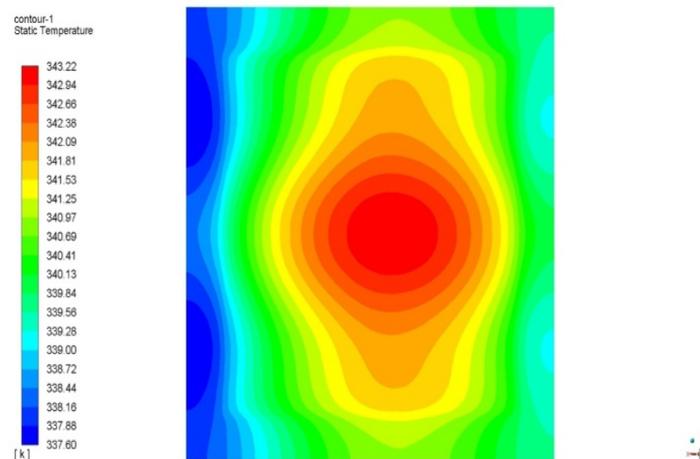


Fig.34. Temperature contour of heat flux wall (Array Water)

International Journal of Innovative Research in Science, Engineering and Technology

(A High Impact Factor, Monthly, Peer Reviewed Journal)

Visit: www.ijirset.com

Vol. 8, Issue 3, March 2019

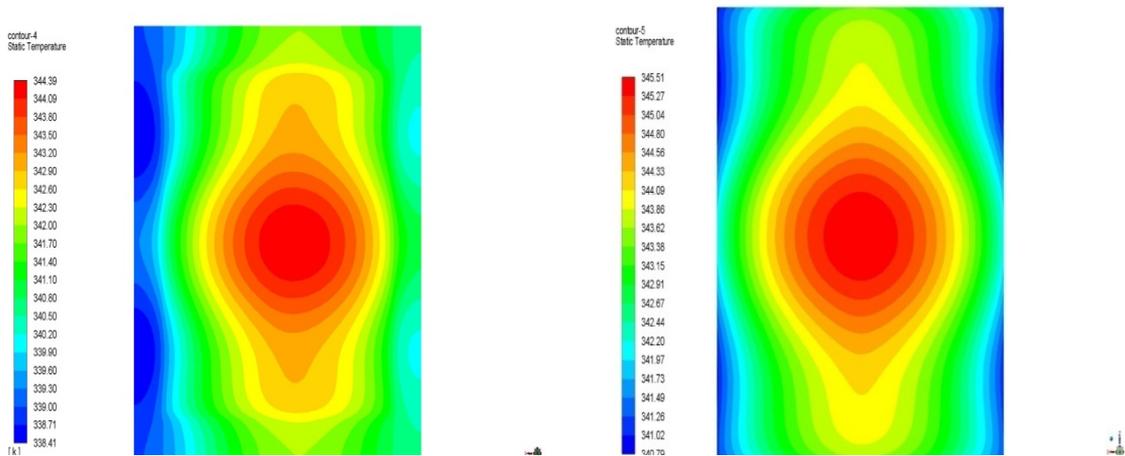


Fig.35. Temperature contour of heat flux wall (ArrayNF1) Fig.36. Temperature contour of heat flux wall (Array NF2)

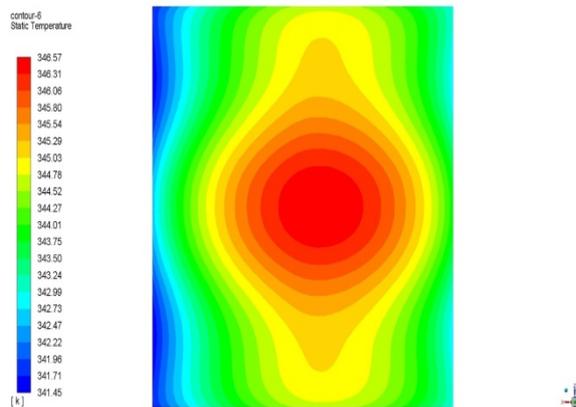


Fig.37. Temperature contour of heat flux wall (Array NF3)

International Journal of Innovative Research in Science, Engineering and Technology

(A High Impact Factor, Monthly, Peer Reviewed Journal)

Visit: www.ijirset.com

Vol. 8, Issue 3, March 2019

Heat Sink Type	Fluid	Pressure		
		Pressure at fluid inlet	Pressure at fluid outlet	Pressure Drop
		P_{in} (Pa)	P_{out} (Pa)	$P_{in} - P_{out}$ (Pa)
Conventional	Water	2.8633	0	2.8633
Array	Water	0.8816	0	0.8816
	NF1	1.0094	0	1.0094
	NF2	1.2278	0	1.2278
	NF3	1.2642	0	1.2642

Heat Sink Type	Fluid	Convective heat transfer coefficient	Average Nusselt Number	Thermal Performance Factor
		h (W/m ² K)	Nu	
Conventional	Water	215.16	373.54	1.000
Array	Water	218.56	369.84	1.466
	NF1	246.02	396.80	1.504
	NF2	241.91	381.55	1.355
	NF3	236.50	361.08	1.269

Performance factor (pf) is obtained from the following expression.

$$pf = \left(\frac{Nu}{Nu^0} \right) / \left(\frac{\Delta p}{\Delta p^0} \right)^{\frac{1}{3}}$$

Where, Nu- Nusselt number of compared model

Nu⁰- Nusselt number of standard model

International Journal of Innovative Research in Science, Engineering and Technology

(A High Impact Factor, Monthly, Peer Reviewed Journal)

Visit: www.ijirset.com

Vol. 8, Issue 3, March 2019

Δp -Pressure drop of compared model

Δp^0 -Pressure drop of standard model

The table below shows that the heat transfer coefficient and Nusselt number of NF1 has the highest value over the other. Performance factor of NF1 is the highest as well. Water has the second highest value of performance factor. NF2 and NF3 lacks in pf value because of their higher viscosity and higher pressure drop characteristics.

VI. CONCLUSION

This project is concerned with the numerical prediction of thermal performance of conventional and array heat sink models with four different heat transfer fluids (Water and Al₂O₃-water nanofluid with 1%, 2% and 3% volume concentrations). From the results obtained it is concluded that the array heat sink model with Alumina-water nanofluid with 1% volume concentration has larger thermal performance factor value than the other models. This is due to the lesser resistance to flow due to lower viscosity than NF2 and NF3. NF1 shows increased convective heat transfer coefficient values, higher Nusselt number value as well. This system is considered as the best model among all models due to its best flow and thermal properties.

REFERENCES

- [1]. Yimin Xuan "Conceptions for heat transfer correlation of nanofluids" IJHMT, 2000
- [2]. S. Suresh "Effect of Al₂O₃-Cu/water hybrid nanofluid in heat transfer" ELSEVIER, 2012
- [3]. K.P. Drummond "A Hierarchical manifold microchannel heatsink array for high-heat-flux two-phase cooling of electronics", CTRC research publications, 2018
- [4]. Y. Raja Shekar, "Heat Transfer with Al₂O₃ nanofluids and twisted pipes for solar thermal applications", ELSEVIER, 2013.
- [5]. Ravishankar Prasad Maurya, Ankesh Kumar Pataskar, "Investigation of Thermal Resistance of Diamond Shaped Pin Fin by CFD" ISSN 2250-2459, ISO 9001:2008 Certified Journal,
- [6]. Behzad Fani, Abbas Abbassi, Mohammad Kalteh, — Effect of nanoparticles size on thermal performance of nanofluid in a trapezoidal microchannel-heat-sink, International Communications in Heat and Mass Transfer, Volume 45, July 2013
- [7]. Han-Taw Chen, Shih-Ting Lai, Li-Ying Haung, — Investigation of heat transfer characteristics in plate-fin heat sink, Applied Thermal Engineering, Volume 50, Issue 1, 10 January 2013