Article Journal-Mathematical and Quantitative Methods
June 2018 Vol.2 No.2 9-19

Numerical analysis of the geometric effects in the optimal performance of a heat sink using nanofluids

Análisis numérico de los efectos geométricos en el rendimiento óptimo de un disipador de calor utilizando nanofluidos

ZUÑIGA-CERROBLANCO, José Luis†*, COLLAZO-BARRIENTOS, Juan Carlos and HERNANDEZ-GUERRERO, Abel

Universidad Politécnica de Juventino Rosas, Calle Hidalgo 102, Comunidad de Valencia, Santa Cruz de Juventino Rosas, Guanajuato, México

ID 1st Author: *José Luis, Zuñiga-Cerroblanco/* **ORC ID:** 0000-0003-0493-8197, **CVU CONACYT ID:** 208410

ID 1st Coauthor: *Juan Carlos, Collazo-Barrientos* / **ORC ID:** 0000-0002-7436-4523, **CVU CONACYT ID:** 208634

ID 2nd Coauthor: *Abel, Hernandez-Guerrero*

Received January 27, 2018; Accepted June 18, 2018

Abstract

In the present work, a thermal numerical analysis of the heat sink is done. The heat sink is used in a cooling process of electronic chips. A geometric configuration for the heat sink is proposed, the fluid is forced to go throw the concentric channels, covering the major area as possible, connections between the channels are added to get different configurations of the heat sink to analyze, and the number of connections analyzed are: 2, 4, 6, 8 and 10. First, water as a working fluid is used, after, Al_2O_3-water nanofluid is analyzed too; the volume concentrations of nanofluid used are: 0.05%, 1% and 3%. In the results, the temperature and pressure contours are reported, as well as the thermal resistance and the pressure drop for the different cases and conditions. The cases analyzed are compared, and the best configuration for the heat sink is obtained. Finally, using the best configuration, a comparison between the water and the nanofluid as a working fluid is done, obtaining that the nanofluid presents an enhance of the heat transfer up to 7 %. Which is an important contribution in the fight against the overheating of electronic chips.

Heat sink, CFD, Nanofluids

Resumen

En el presente trabajo se realiza un análisis numérico del desempeño de un disipador de calor, el cual es usado en para el enfriamiento de chips electrónicos. Se propone una geometría para el disipador de calor, la cual consiste en forzar al flujo a fluir por canales concéntricos, cubriendo la mayor cantidad de área posible, se realizan modificaciones a la geometría añadiendo conexiones entre los canales de flujo y determinando si esto contribuye al aumento de transferencia de calor; el número de conexiones entre los canales que se analizan son: 2, 4, 6, 8 y 10. En un principio se utiliza agua como fluido de enfriamiento, posteriormente se analiza el disipador de calor utilizando un nanofluido, el cual es compuesto de nanopartículas de alumina (Al_2O_3) a concentraciones volumétricas de 0.05%, 1% y 3%. En los resultados se reportan los contornos de temperatura y presión para los diferentes casos de análisis, así como también la resistencia térmica y la caída de presión. Se comparan los casos analizados y se obtiene una configuración del disipador de calor que tenga un mejor desempeño térmico. Finalmente utilizando la mejor configuración, se hace una comparación de la resistencia térmica y la caída de presión usando agua y el nanofluido, obtenido que el nanofluido presenta un aumento en la transferencia de calor hasta del 7 %.

Disipador de calor, CFD, nanofluidos

Citation: ZUÑIGA-CERROBLANCO, José Luis, COLLAZO-BARRIENTOS, Juan Carlos and HERNANDEZ-GUERRERO, Abel. Numerical analysis of the geometric effects in the optimal performance of a heat sink using nanofluids. Journal-Mathematical and Quantitative Methods. 2018. 2-2: 9-19.

---

* Correspondence to Author (email: jzunigac_ptc@upjr.edu.mx)

† Researcher contributing first author.

Introduction

The technological advances in the area of the electronic industry have marked the life of man and are currently a very useful tool that accompanies the daily life of people of all ages; For example, today there are cell phones and computers that just a few decades ago it was difficult to imagine that they had such a degree of development.

These advances have generated areas of opportunity for improvement, in which we can work to ensure the proper functioning of electronic devices. One of these areas, which must be addressed is the cooling of electronic devices. Unfortunately, the technological advances have overcome the cooling techniques, in this way, the overheating of the electronic chips causes a bad performance and sometimes very frequent until the total damage of the same.

Due to the above, different cooling techniques have been proposed to meet the needs of the electronics industry. These techniques mainly use conduction and convection heat transfer modes, occupying air-cooled heat sinks as one of the main solutions, due to its low cost and easy installation. However, this technique has reached its design limits, and for the demands of the heat generated by the chips of today are not enough, that is why we have opted for the use of heat sinks using water as cooling fluid [1], this technique offers a higher coefficient of heat transfer, which is why it is being used in the electronics industry.

Recently the use of nanoparticles in the cooling water of the heat sinks has been incorporated, thus forming a nanofluid, which serves to improve the thermophysical properties and consequently, increase the thermal performance of the heat sinks.

In the present work a design of a heat sink for the cooling of electronic chips is proposed and analyzed, in which water and nanofluids will be used as cooling fluid.

Background

The technique of cooling by liquid is not a new technique, this has been used since the decade of the 80s and takes popularity today because it offers higher heat transfer coefficients than using air as a cooling fluid.

Of the first researchers who used this technique, we have Tuckerman and Pease [2], who in 1981 worked on the dissipation of heat using microchannels and water as a cooling fluid, they managed to remove up to 790 W / cm^2 of constant heat flux over the surface of the heatsink, having a thermal resistance in the dissipator of 0.09 cm^{2}K/W.

In order to improve the heat dissipation, different types and configurations of heat sinks have been proposed, for example, channels with rectangular cross section [2-3], circular [4], triangular [5] and trapezoidal [6]; some other researchers have used fins inside the channels and have varied the parameters of said fins [7-8], this in order to look for the best configurations for the dissipators.

All the modifications proposed by the different authors are with the intention of increasing the heat removed and also having a uniform temperature distribution, this to avoid having hot spots that can damage the electronic chips.

Rubio-Jiménez et al. [7] numerically analyze a finned heat sink, investigate the effect on the number of Nusselt due to the modification of the geometric parameters of the fins, as well as the arrangement of the fins and the number of these. In their reported results they find an optimal configuration for the fin geometry. Additionally, the density of the fins varies in different sections of the channel. Obtaining in this way a better thermal performance of the heatsink.

Upadhye et al. [9] numerically investigate the direct cooling of an electronic chip. They analyze it based on the variation of the channel geometry. Their results indicate that using narrower channels, the heat transfer of the dissipater is considerably improved.

Another of the numerical analyzes found in the literature is the fact by Li et al. [10], who investigate the influence of the geometric parameters of the channel and the thermophysical properties in the heat transfer of a dissipator. In their reported results, they obtain the detailed behavior of the heat transfer coefficient and a correlation for the global Nusselt number.

In addition to the numerical analyzes, experimental analyzes have been developed. In this area there are works such as the one carried out by Wei et al [11], who experimentally analyze the performance of a dissipator. They use different configurations for the assembly of the heat sink with the electronic circuit. They also evaluate the effects on thermal performance due to the direction of the cooling flow. They report heat transfer and pressure drop. With these results they obtain an optimal design of the assembly and direction of the cooling flow.

Another of the experimental works is the fact by Qu et al. [12], they carry out the characterization of pressure drop and heat transfer on a heatsink. They consider rectangular channels and handle a wide range of heat flow supplied by the heatsink. They obtain numerical and experimental results, they demonstrate that by using the Navier-Stokes equations, the continuity equation and the energy equation, the behavior of the fluid flow and its heat transfer can be predicted with great accuracy.

As a result of the constant search for better and more efficient techniques in the removal of heat from electronic chips, the use of nanofluids [13] and [14] has been incorporated. A nanofluid is a mixture of nanoscale particles with a base fluid [15]; different types of metallic elements can be used as nanoparticles, among which are commonly used, gold, silver, alumina and titanium dioxide; The objective of the use of nanoparticles is to improve the thermophysical properties of the fluid, such as: thermal conductivity, density, viscosity and the convective heat transfer coefficient [16], [17] and [18].

Pantzali et al. [19] investigated the use of CuO nanoparticles suspended in water as a nanofluid and tested them on a plate heat exchanger.

The experiments confirm that the use of the nanofluid inside the exchanger improves the heat transfer for the turbulent regime, however, for the laminar regime, the increase in viscosity affects the instability of the suspension.

Shafahi et al. [20] conducted an analysis on the thermal performance of a heat pipe using nanofluids. They analyzed different types of nanoparticles (Al_2O_3, CuO and TiO) in the working fluid. Within their results they report, the thermal resistance, the temperature distribution and the maximum capillary heat transfer, as well as the effect of the size of the nanoparticles; the results obtained show that the nanoparticles in the fluid improve the thermal performance of the heat pipe, reducing the thermal resistance.

The size of the nanoparticles has been investigated by Davarnejad et al. [21], who analyzed the thermal performance for a particular size of Al2O3 of 20 nm and 50 nm; the results show that the nanofluid with particles of 20 nm has a higher heat transfer coefficient compared to the 50 nm particles. A similar work was done by Moraveji et al. [22], their results show the same behavior for the heat transfer coefficient compared with the results reported in the technical literature; based on their results they propose a correlation for the Nuselt number, which presents a good precision compared with experimental data.

Geometry

To propose the configuration of the heat sink, an Intel Core i7 processor [23] is considered, which has a thermal design power of 35 W. This power is calculated in an operation with all the active cores and in a high demand defined by Intel. The temperature of the processor at rest must be between 34 ° C and 39 ° C, the normal working temperature is between 55 ° C and 65 ° C, the maximum temperature supported by the processor is 72 ° C, work should be avoided or exceed this temperature, otherwise the processor may suffer some irreparable damage.

This is why you must avoid overheating and the creation of hot spots on the surface of the electronic chip. The installation of a heat sink on the surface of the processor, allows to keep the processor at a suitable temperature during operation, the heatsink must be easy to install and must be able to do any person.

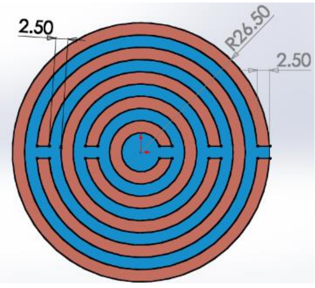


Figure 1 Top view of the proposed heatsink geometry, units in mm

Figure 1 shows the proposed configuration for the heat sink in the present investigation. This shows a channel that goes around the surface of the base of the heatsink that is in direct contact with the hot surface of the processor. Water enters the center of the heatsink and exits at one end of the heatsink, as shown in Figure 2.

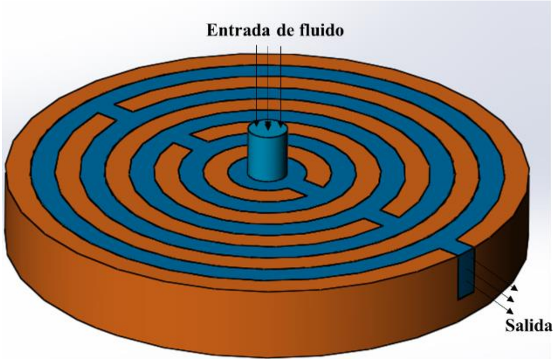
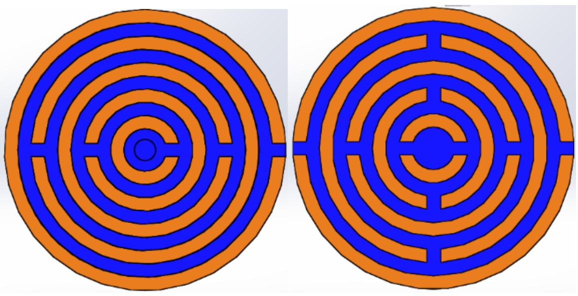
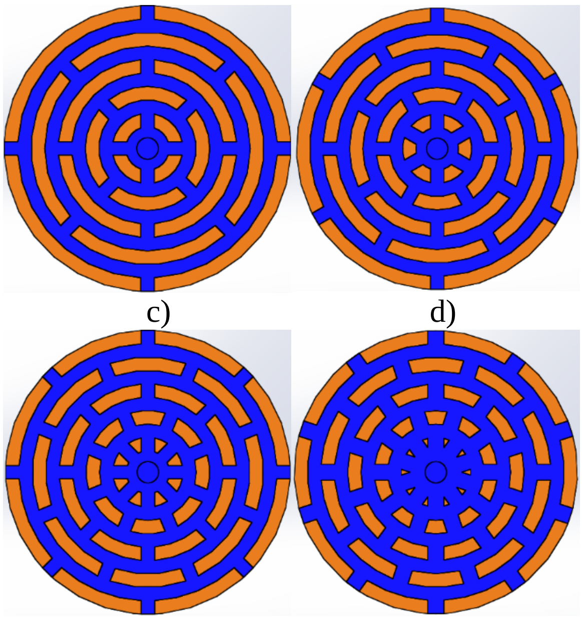


Figure 2 Input and output of the working fluid in the heat sink

Positioning the water inlet and outlet in the aforementioned way, will cause the water to evenly cool the surface of the processor, avoiding hot spots, this also helps the installation of the hoses, thinking about future work where the experimental tests of the heat sink are carried out. The walls of the channels will work like fins, increasing the area of heat transfer between the material and the water.



a) b)



c) d)

e) f)

Figure 3 Top view of the different geometries that are analyzed. a) 1 connection, b) 2 connections, c) 4 connections, d) 6 connections, e) 8 connections and f) 10 connections.

In Figure 3, the 6 different configurations that are analyzed numerically are shown, in each of them the number of connections between the concentric channels is increased, thus having the following configurations: a) 1 connection, b) 2 connections, c) 4 connections, d) 6 connections, e) 8 connections and f) 10 connections. What is sought by increasing the number of connections is that the flow is distributed radially and observe the impact that this type of flow has on the thermal and hydrodynamic performance of the heat sink. It is observed that with the increase in the number of connections the number of outlets of the working fluid also increases.

Table 1 presents the analyzed cases, for each case there is a different number of connections between the channels, the test speeds for each case are 0.1 m / s, 0.2 m / s, 0.3 m / s and 0.5 m / s.

Caso	**Número de conexiones entre el canal.**
1	1
2	2
3	4
4	6
5	8
6	10

Table 1 Cases studied in the present work

Once the different geometric configurations of the heat sink have been defined, the computational model is used to solve the system governing equations and obtain the results of heat transfer and water flow behavior.

Computational model

The development of the computational model begins with the realization of the geometry in SolidWorks, the different cases of analysis shown in Table 1 are created. In Figure 4 one of the geometries made for the heat sink is shown, the part can be observed solid and the part of the fluid, as well as the entrance and exits.

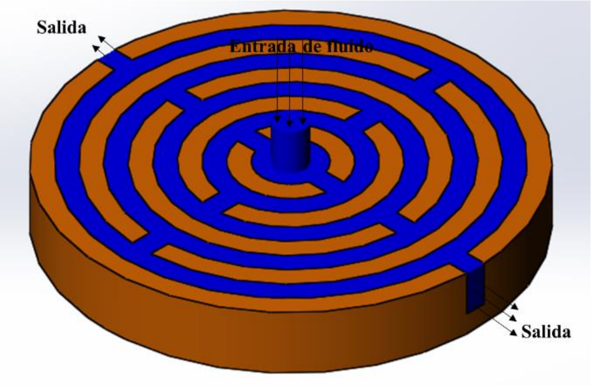


Figure 4 Isometric view of the heat sink, Case 2.

Mesh analysis

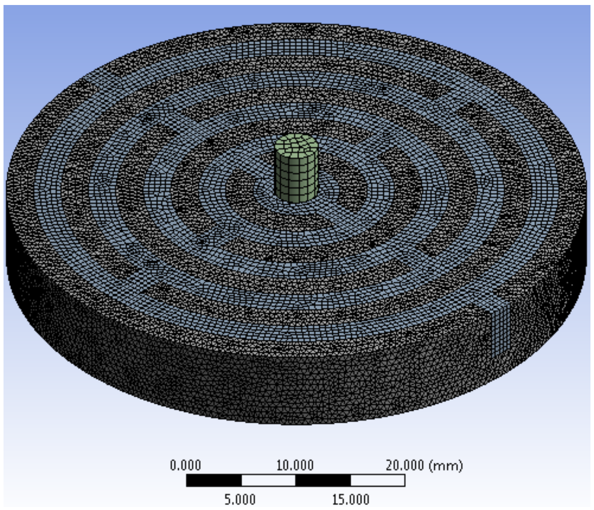


Figure 5 Mesh made on the dissipater geometry, Case 2.

Once the geometric models of the different case studies have been built, the. IGES protocol is used to export the geometry to the ANSYS software, where the mesh is constructed, which will make possible the solution of the numerical model.

Figure 5 shows the mesh constructed for the domain of the solid and water. It starts with a coarse mesh, and it is refined until a dense and better-quality mesh is obtained. A mesh sensitivity analysis is performed, this analysis guarantees that the results obtained will not suffer variations due to the quality of the mesh.

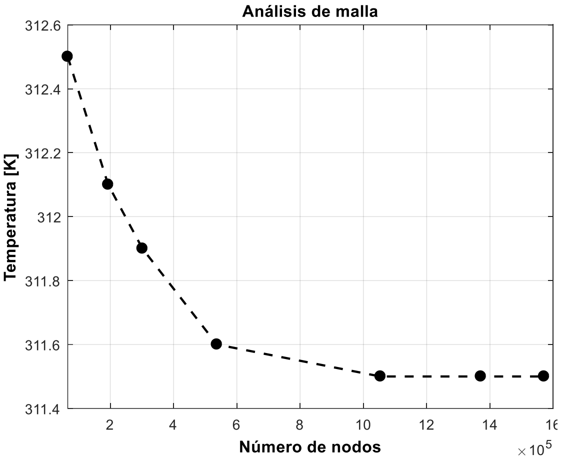


Figure 6 Variation of the surface temperature of the dissipator for different numbers of mesh nodes

Figure 6 shows the mesh analysis performed, in the graph we have the temperature taken in a specific area of the heat sink once the convergence criteria have been met, which is recorded for different numbers of nodes used in the mesh, it is observed that the temperature undergoes a change when the numbers of nodes in the mesh increase, no longer undergoing any change from a number of nodes of 1054092, thus obtaining the optimum mesh for our numerical model.

Border conditions

The next step is to assign the boundary conditions for the solution of the governing equations.

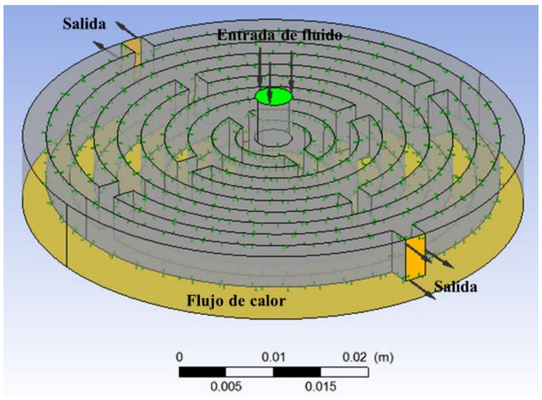


Figure 7 Boundary conditions for the numerical model

Figure 7 contains the boundary conditions assigned to the numerical model, an inlet condition for the cooling water can be observed, as well as an exit condition. At the base of the heatsink the constant heat flow condition is assigned, which simulates the heat generated by the operation of the processor. The walls of the channel are considered to be non-slip, and the remaining walls of the heat sink are considered as adiabatic walls.

In addition to the boundary conditions certain conditions are taken to simplify the numerical analysis, the conditions taken are the following:

Constant thermophysical properties.

The water temperature at the entrance of the heat sink channel is 293 K.

The heat flow at the base of the heatsink is 35 W.

Stable state for heat transfer.

Turbulent flow.

Turbulence model k-ϵ (RNG).

The heat transfer mode by radiation is neglected.

It is also considered that the heat sink is copper and the cooling fluid is water. Table 2 shows the thermophysical properties considered, both for copper and for water.

Property **Water**	**Value**
Thermal conductivity (W / m-K)	0.5948
Specific heat (kJ / kg-K)	4.183
Density (kg / m3)	997.1
Viscosity (kg / m-s)	0.0008905
Copper	
Thermal conductivity (W / m-K)	387.6
Specific heat (kJ / kg-K)	0.381
Density (kg / m3)	8978

Table 2 Thermophysical properties of copper and water

Governing equations

Based on the phenomenon that is required to analyze, we have that the governing equations for the domain of the fluid are: the equations of conservation of mass, equation (1), for now, equations (2-4) and energy, equation (5)).

$$\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y}+\frac{\partial w}{\partial z}=0 \tag{1}$$

$$\rho_f\left(u\frac{\partial u}{\partial x}+v\frac{\partial u}{\partial y}+w\frac{\partial u}{\partial z}\right)=-\frac{\partial p}{\partial x}+\mu\nabla^2 u \tag{2}$$

$$\rho_{\mathrm{f}}\left(u\frac{\partial v}{\partial x}+v\frac{\partial v}{\partial y}+w\frac{\partial v}{\partial z}\right)=-\frac{\partial p}{\partial y}+\mu\nabla^2 v \tag{3}$$

$$\rho_{\mathrm{f}}\left(u\frac{\partial w}{\partial x}+v\frac{\partial w}{\partial y}+w\frac{\partial w}{\partial z}\right)=-\frac{\partial p}{\partial z}+\mu\nabla^2 w \tag{4}$$

$$\rho_{\mathrm{f}}c_{\mathrm{pf}}\left(u\frac{\partial T}{\partial x}+v\frac{\partial T}{\partial y}+w\frac{\partial T}{\partial z}\right)=k_{\mathrm{f}}\nabla^2 T \tag{5}$$

For the solid domain we have that the equation of the energy is like the equation (6):

$$\nabla^2 T=0 \tag{6}$$

It can be used as a parameter of comparison between the different cases of study to the heat resistance of the heat sink, this can be defined as a function of the local temperature of the surface, minus the temperature at which the fluid enters, this divided into in heat flow supplied to the bottom of the heat sink.

$$R_t=\frac{T_{suf}-T_{ent,f}}{q''} \tag{7}$$

Nanofluids

The thermophysical properties of nanofluids such as density, specific heat, viscosity and thermal conductivity must be calculated. In the technical literature there are correlations for internal flow that can be used for the calculation of these properties [24]. Equation (8) and (9) determine density and specific heat for multicomponent substances.

$$\rho_{nf}=(1-\varphi)\rho_{fb}+\varphi\rho_p \tag{8}$$

$$Cp_{nf} = (1-\varphi)Cp_{fb} + \varphi Cp_p \tag{9}$$

The thermal conductivity of a nanofluid can be determined using equation (10), which is proposed by Maxwell [15]; In his work, Maxwell considers the phenomenon of thermal conduction in a diluted suspension of spherical particles, neglecting the interactions between particles.

$$C_{ef} = k_{fb} + 3\varphi_p \frac{k_p - k_{fb}}{2k_{fb} + k_p - \varphi_p (k_p - k_{fb})} k_{fb} \tag{10}$$

Where k represents the thermal conductivit,, volume fraction of the nanoparticle and the subscripts, and mean, effective, particle and base fluid, respectively.

Another relevant property is the viscosity of the nanofluid. Batchelor [25] proposed a model for the viscosity of nanofluids, this model, equation (11) is derived from previously proposed models.

$$\mu_{ef} = \mu_{fb}\left(1 + 2.5\varphi_p + 6.5\varphi_p^2\right) \tag{11}$$

Property	Al_2O_3	Water	Nanofluid φ=0.5 %	φ=1 %	φ=3 %
k $(Wm^{-2}k^{-1})$	40	0.5948	0.6034	0.612	0.6475
ρ (kgm^{-3})	3860	997.1	1011.41	1025.73	1082.98
Cp $(Jkg^{-1}K^{-1})$	849	4183.1	4166.43	4149.759	4083.07
$\mu(x10^{-4})$ $(mPa*s)$		8.905	9.018	9.133	9.625

Table 3 Thermophysical properties of nanofluids for different concentration volumes

The above equations are used to determine the modified properties of the nanofluid used in this work. Table 3 shows the properties of nanofluids for different concentration volumes of nanoparticles. It is advisable not to use a concentration of more than 3% in a nanofluid.

Results

In this section we report the results obtained from the numerical solution of the governing equations under the boundary conditions described above. The thermal results include temperature contours for different analyzed cases, as well as the thermal resistance, which helps us to know the thermal performance of each case analyzed.

On the other hand, hydrodynamic results are also reported, which include the pressure drop as well as the current flow lines.

Thermal

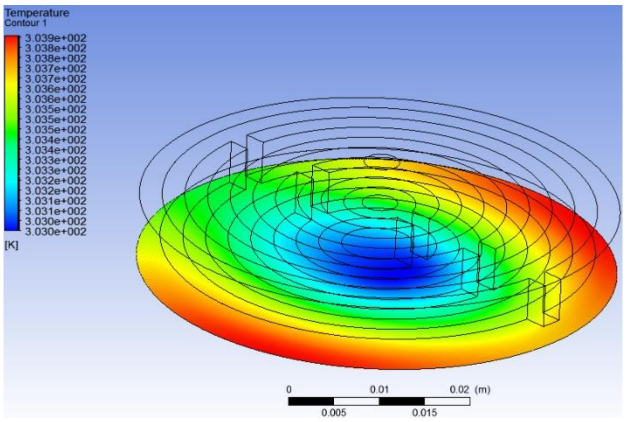


Figure 8 Temperature contours for Case 1, with an entry speed of 0.3 m / s

Figure 8 shows the temperature contour for Case 1, with a water inlet velocity of 0.3 m / s. It can be seen in the image that the higher temperature is 303 K, it is within the resting temperature range of the processor. Although a temperature variation is observed, this is minimal, around 1 ° C, so it can be said that there is a uniform distribution of temperature on the base of the dissipator, which is the one that would be in direct contact with the processor.

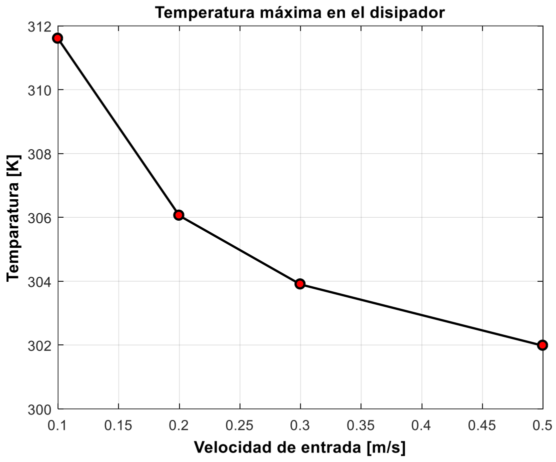


Figure 9 Maximum temperature for the heat sink of Case 1 at different analysis speeds

Figure 9 shows the maximum temperature at the base of the heatsink for Case 1 and for the water inlet velocities of 0.1 m / s, 0.2 m / s, 0.3 m / s and 0.5 m / s; As expected at higher input speed the maximum temperature decreases, this due to the increase in the heat transfer coefficient. The range of maximum temperatures reached for this case is within the normal temperature range for the processor.

Figure 10 shows the maximum temperatures for each case study, with a water inlet velocity of 0.1 m / s. It is observed that with the increase of connections between the channels, the maximum temperature increases. Having a higher temperature for Case 6, which is where you have 10 connections between the channels. In none of the cases is the temperature range exceeded for the proper functioning of the processor, in fact the temperatures obtained for the different cases studied are always below this range.

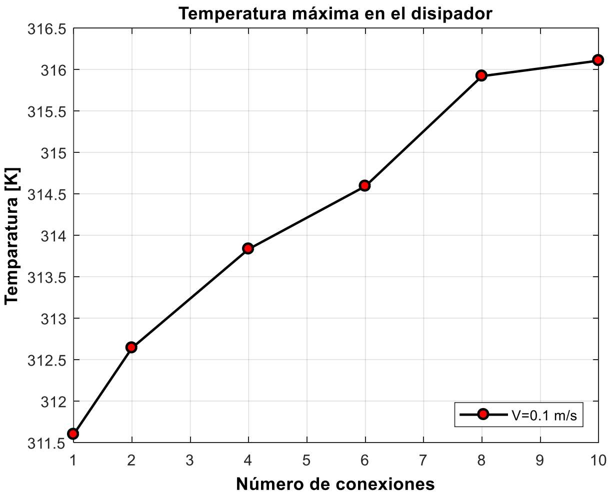


Figure 10 Maximum temperature for the heat sink for all analysis cases, at a water inlet velocity of 0.1 m / s

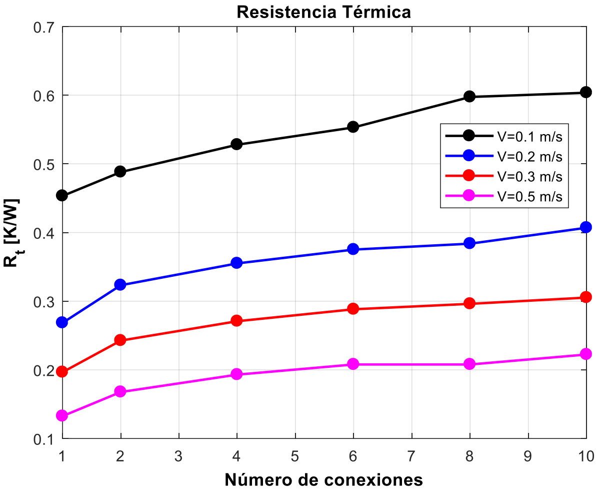


Figure 11 Thermal resistance vs the number of connections between the channels, for the different fluid inlet speeds

Figure 11 shows the resistance for each of the study cases for water inlet velocities of 0.1 m / s, 0.2 m / s, 0.3 m / s and 0.5 m / s. It is observed that the thermal resistance increases with the number of connections between the channels, this is because the flow flows radially and not concentric with the increase of the connections.

Figure 12 shows the comparison of thermal resistance for the use of water and nanofluid. The comparison is made with the configuration of the heat sink of Case 2, and for a flow inlet velocity of 0.3 m / s. It is observed that the thermal resistance is lower when using the nanofluid compared to water, and that a higher concentration of the nanofluid gives a lower thermal residence. The reduction of thermal resistance with the use of nanofluid is around 7%.

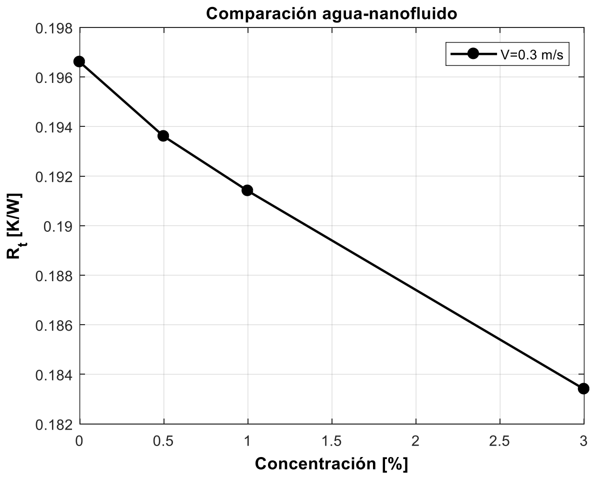


Figure 12 Comparison of the thermal resistance of water and nanofluid at different concentrations and an entry speed of 0.3 m / s

Hydrodynamics

Figure 13 shows the pressure contour for Case 1 at an input speed of 0.3 m / s. As expected, a higher pressure is obtained at the inlet of the fluid, and it decreases as the fluid approaches the outlet. For this case, a greater energy is required to make the fluid circulate through the channels, this is due to the fact that there is only one outlet and necessarily the total of the fluid that enters must exit through it.

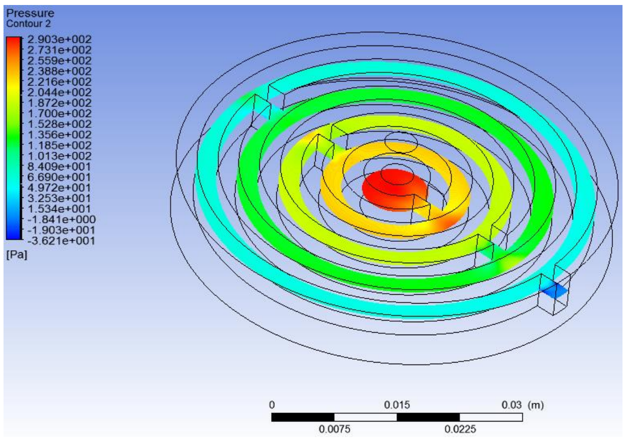


Figure 13 Pressure contours for Case 1, with an entry speed of 0.3 m / s

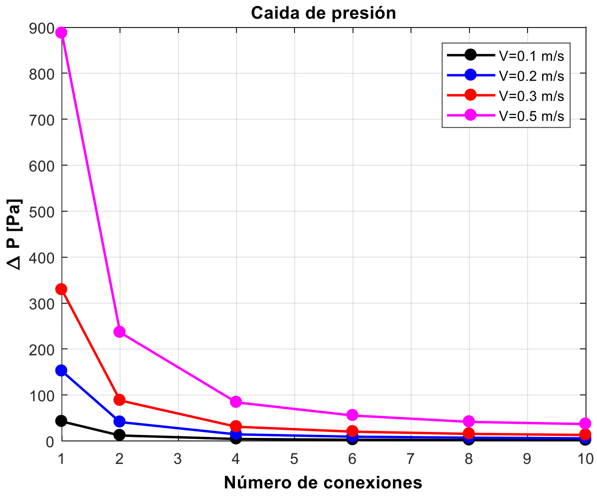


Figure 14 Pressure drop vs the number of connections between the channels, for the different fluid inlet velocities

Making a comparison of the different pressure drops for each of the study cases and for the speeds considered, the results shown in Figure 14 are obtained, in which it is observed that there is a greater pressure drop for Case 1 compared with the rest of the cases, and this increases with the increase in speed. A higher pressure drop requires more energy to move the cooling fluid.

Figure 15 shows the current lines following the flow for Case 2 at an input speed of 0.1 m / s. It is observed how the flow covers the total of the area of the channel going to the outputs that are at the ends of the geometry.

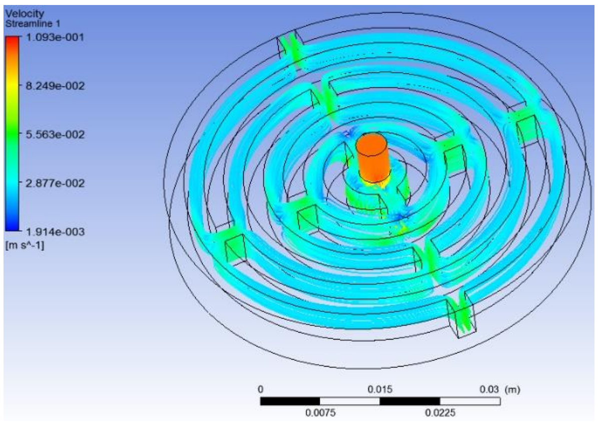


Figure 15 Current lines for Case 2 at an input speed of 0.1 m / s

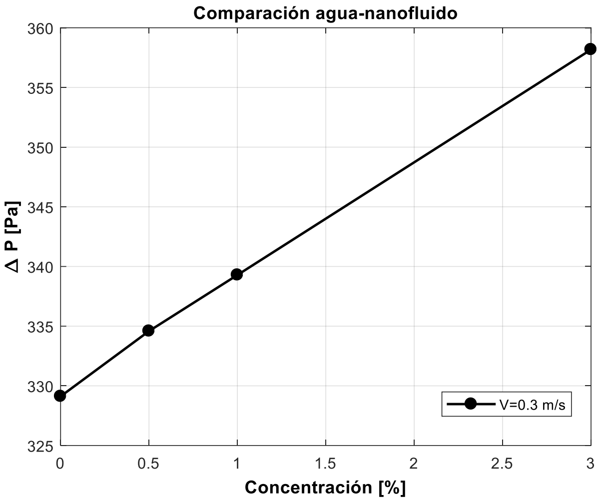


Figure 16 Comparison of the pressure drop between water and nanofluid at different concentrations and an entry speed of 0.3 m / s

Figure 16 shows the comparison of the pressure drop for the use of water and the nanofluid. The comparison is made with the configuration of the heat sink of Case 2, and for a flow inlet velocity of 0.3 m / s. An increase in pressure drop is observed with the use of nanofluid and with the increase in the concentration of this.

Conclusions

A numerical analysis of a heat sink is performed, which can be applied in cooling in the electronics industry. The different configurations of the heat sink geometry are constructed, this geometry is mesh and a mesh sensitivity analysis is performed.

Through specialized software, the governing equations are resolved with the boundary conditions and the indicated considerations.

The results of the thermal and hydraulic performance are obtained, for the different configurations of the dissipators and different fluid inlet velocities. From the results it is observed that the heat sink has a good performance, because in the case of the processor being analyzed, the temperature of the surface of the dissipator is kept below the acceptable range for its correct operation.

Regarding the different configurations analyzed, it is obtained that the thermal resistance increases and the pressure drop decreases with the increase of connections between the channels, hence the configuration of Case 2 can be chosen, which has two connections between the two channels, with this there is a slight increase in the thermal resistance for a considerable decrease in the pressure drop compared with Case 1, which requires less energy for the fluid to be circulating through the dissipator. In the other cases in which the connections are increased, the thermal resistance increases and the pressure drop no longer has considerable decreases compared to Case 2. In order to choose an optimal case, an analysis of minimum generation of entropy must be carried out, where both thermal and hydrodynamic effects are considered in the same objective function to be optimized, since the configuration with the least thermal resistance can be chosen, but this also has the highest pressure drop of all cases, which is why a function that considers both effects is necessary, this will be done in future works.

It is verified that the use of nanofluids helps to increase the heat transfer, obtaining up to 7% in the reduction of the thermal resistance when Al2O3-water is used as a nanofluid in a concentration of 3%. Therefore, the use of a nanofluid as a working fluid is a viable option for the cooling of electronic devices that use this type of dissipaters.

As future work, heat sinks will be manufactured and experimental tests will be carried out to compare them with the numerical results. Then proceed to perform an optimization of the geometric parameters by the method of minimum generation of entropy.

Nomenclature

k	Thermal conductivity, (W / m-K)
c_p	Specific heat, (kJ / kg-K)
q''	Heat flow, (W / m2)
Rt	Thermal resistance, (K / W)
T	Temperature, (K)
ΔP	Pressure drop, (Pa)
u	Component of the speed in the x direction, (m / s)
v	Component of the speed in the direction y, (m / s)
w	Component of the speed in the z direction, (m / s)
	Greek symbols
ρ	Density, (kg / m^3)
φ	Volume concentration,%
μ	Dynamic viscosity, (kg / m-s)
	Subscript
ef	Effective
fb	Base fluid
p	Particle
nf	Nanofluid
ent	Entry

References

S.G. Kandlikar. (2005) High Flux Heat Removal with Microchannels –A Roadmap of Challenges and Opportunities. Heat Transfer Engineering. 26 (8). 5-14.

D.B. Tuckerman, R.F. Pease. (1981). High-Performance Heat Sinking for VLSI. IEEE Electron Devices Lett. EDL-2 (5). 126-129.

M. M. Mohamed, M.A. Abd El-Baky. (2013). Air Cooling of Mini-Channel Heat Sink in Electronic Devices. Journal of Electronics Cooling and Thermal Control. 3. 49-57.

M.R. Sohel, R. Saidur, Mohd Faizul Mohd Sabri, M. Kamalisarvestan, M.M Elias, Ali Ijam. (2013). Investigating the Heat Transfer Performance and Thermophysical Properties of Nanofluids in a Circular Microchannel. International Communications in Heat and Mass Transfer. 42. 75-81.

S.Z Heris, S.H. Noie, E. Talaii, J. Sargolzaei. (2011). Numerical Investigation of Al2O3/water Nanofluid Laminar Convective Heat Transfer through Triangular Ducts. Nanoscale Research Letters. 6.

A. Husain, K.Y. Kim. (2009). Thermal Optimization of a Microchannel Heat Sink with Trapezoidal Cross Section. Journal of Electronic Packaging. ASME. 31. 2.

C.A. Rubio Jimenez. S.G. Kandilikar, A. Hernandez Guerrero. (2012). Numerical Analysis of Novel Micro Pin Fin Heat Sink with Variable Fin Density. IEEE Trans. Compon. Packag. Manuf. 2 (5). 825-833.

D. Lorenzini-Gutierrez. S.G.Kandlikar. (2004). Variable Fin Density Flow Channels for Effective Cooling and Mitigation of Temperature Nonunoformity in Tree-Dimensional Integrated Circuits. Journal of Electronic Packaging. Microchannels and Minichannels. 136.

H. R. Upadhye., S.G. Kandlikar. (2004), Optimization of Microchannel Geometry for Direct Chip Cooling Using Single Phase Heat Transfer. ASME. International Conference on Nanochannels, Microchannels, and Minichannels. 679-6685.

J. Li., G.P. Peterson., P. Cheng. (2004). Three-Dimensional Analysis of Heat Transfer in a Micro-Heat Sink with Single Phase Flow. International Journal of Heat and Mas Transfer. 47. 19-20.

X. Wei, Y. Joshi, M.K. Patterson. (2007). Experimental and Numerical Study of a Stacked Microchannel Heat Sink for Liquid Cooling of Microelectronic Devices. Journal of Heat Transfer. ASME. 129.

W. Qu., I. Mundawar. (2002). Experimental and Numerical Study of Pressure Drop and Heat Transfer in a Single-Phase Micro-Channel Heat Sink. International Journal of Heat and Mas Transfer. 45. 12.

S. U.S. Choi. (2009). Nanofluids: From Vision to Reality through Research. ASME J. Heat Transfer. 131. 033106.

G. Huminic, and A. Huminic. (2012). Applications of nanofluids in heat exchanger: A Review. Renewable and Sustainable Energy Reviews. 16. 5625-5638.

Maxwell JC. (1873). Treatise on Electricity and magnetism. Oxford: Clarendon Press.

Lazarus Godson, B. Raja, D. Mohan Lal, S. Wongwises. (2010). Enhancement of Heat Transfer Using Nanofluids— An Overview. Renewable and Sustainable Energy Reviews 14. 629–641.

Khalil Khanafer, Kambiz Vafai. (2011). A Critical Synthesis of Thermophysical Characteristics of Nanofluids. International Journal of Heat and Mass Transfer. 54. 4410–4428.

Xiang-Qi Wang, Arun S. Mujumdar. (2007). Heat Transfer Characteristics of Nanofluids: A Review. International Journal of Thermal Sciences. 46. 1–19.

M. N. Pantzali, A. A. Mouza, S. V. Paras. (2009). Investigating the Efficacy of Nanofluids as Coolants in Plate Heat Exchangers (PHE). Chemical Engineering Science. 64. 3290–3300.

Maryam Shafahi, Vincenzo Bianco, Kambiz Vafai, Oronzio Manca. (2010). An Investigation of the Thermal Performance of Cylindrical Heat Pipes Using Nanofluids. International Journal of Heat and Mass Transfer. 53. 376–383.

Reza Davarnejad, Sara Barati and Maryam Kooshki. (2013). CFD Simulation of the Effect of Particle Size on the Nanofluids Convective Heat Transfer in the Developed Region in a Circular Tube. SpringerPlus. 2:192. doi:10.1186/2193-1801-2-192.

Mostafa Keshavarz Moraveji, Mehdi Darabi, Seyyed Mohammad Hossein Haddad, Reza Davarnejad. (2011). Modeling of Convective Heat Transfer of a Nanofluid in the Developing Region of Tube Flow with Computational Fluid Dynamics. International Communications in Heat and Mass Transfer. 38. 1291–1295.

Intel. (s.f.). Intel Latinoamérica. (2017).

Yimin Xuan, Wilfried Roetzel. (2000). Conceptions for Heat Transfer Correlation of Nanofluids. InternationalJournal of Heat and Mass Transfer. 43. 3701-3707.

Batchelor, G.K. (1977). The Effect of Brownian Motion on the Bulk Stress in a Suspension of Spherical Particles. Journal of Fluid Mechanics. 83. 97-117. doi:10.1017/S0022112077001062