https://doi.org/10.31436/iiumej.v23i1.2113

COMPUTATIONAL FLUID DYNAMICS MODELING OF MICROCHANNELS COOLING FOR ELECTRONIC MICRODEVICES

JONATHAN FÁBREGAS [1], HENRY SANTAMARÍA[1], EDGARDO BUELVAS[1], SAUL PÉREZ[1], CARLOS DÍAZ[1], JAVIER CARPINTERO[2], RICARDO MENDOZA[3] AND JENNIFER VILLA[2*]

[1]*Faculty of Engineering, Universidad Autónoma del Caribe, Colombia*
[2]*Department of Civil and Environmental, Universidad De La Costa, Calle 58 #55-66, 080002 Barranquilla, Atlántico, Colombia*
[2]*Energy Department, Universidad De La Costa, Calle 58 #55-66, 080002 Barranquilla, Atlántico, Colombia*

[*] Corresponding author: jvilla27@cuc.edu.co

(Received: 13th June 2021; Accepted: 13th October 2021; Published on-line: 4th January 2022)

ABSTRACT: A simulation of the cooling of electronic devices was carried out by means of microchannels, using water as a coolant to dissipate the heat generated from a computer processor, and thus stabilize its optimum operating temperature. For the development of this study, computational fluid mechanics modeling was established in order to determine the temperature profiles, pressure profiles, and velocity behavior of the working fluid in the microchannel. In the results of the study, the operating temperatures of the computer processor were obtained, in the ranges of 303 K to 307 K, with fluid velocities in the microchannels of 5 m/s, a pressure drop of 633.7 kPa, and a factor of safety of the design of the microchannel of 15. From the results, the improvement of the heat transfer in a cooling system of electronic devices was evidenced when using a coolant as a working fluid compared to the cooling by forced air flow traditional.

ABSTRAK: Simulasi penyejukan alatan elektronik telah dibina menggunakan saluran mikro, di samping air sebagai agen penyejuk bagi menghilangkan haba yang terhasil dari pemproses komputer, dan penstabil pada suhu operasi optimum. Kajian ini mengenai model komputasi mekanik bendalir bagi menentukan profil suhu, profil tekanan, dan halaju perubahan bendalir dalam saluran mikro. Dapatan kajian menunjukkan suhu operasi pemproses komputer adalah pada julat suhu 303 K sehingga 307 K, dengan halaju bendalir dalam saluran mikro adalah pada kelajuan 5 m/s, penurunan tekanan sebanyak 633.7 kPa, dan faktor keselamatan 15 bagi reka bentuk saluran mikro. Ini menunjukkan terdapat kenaikan pemindahan haba bagi sistem penyejukan alatan elektronik ini, terutama apabila cecair digunakan sebagai penyejuk haba berbanding kaedah tradisi iaitu dengan mengguna pakai aliran udara sebagai agen penyejuk.

KEYWORDS: *computational fluids dynamics; microchannels; processor; cooling*

1. INTRODUCTION

At present, there is a growing demand for greater performance in electronic devices and for reliability in the level of heat rejection, which make it necessary to create alternatives for the removal of heat and which in turn are much more efficient. Therefore, some alternatives have been presented to solve this problem by developing research using

https://doi.org/10.31436/iiumej.v23i1.2113

microchannels (MC). Belhardj et al. [1] used MC with a phase change fluid to improve heat transfer in the cooling of processors.

Chen and Ding [2] analyzed the performance of cooling devices using MC with working fluids of water and nanofluids. Their results showed adequate behavior with the two proposed alternatives. Moreover, Chiu et al. [3] studied the effect of the MC geometry and the pressure used in the heat transfer. It was found that a higher pressure drop improved the heat transfer coefficient. Bosi et al. [4] designed a series of MC modules with different channel separation configurations to study the cooling phenomenon of a silicon pixel detector system. In addition, Brinda et al. [5], in their study, propose a series of MC in a stepped form with a rectangular cross-section to increase the area of the walls, which decreases the thermal resistance and increases the heat transfer coefficient.

Naqiuddin et al. [6] carried out a study applying the technology of MC for the cooling of electronic chips, adapting the principles of straight channels by a segmented method. Modeling was developed for this research by computational fluid dynamics (CFD) and the statistical method of Taguchi-gray to optimize the system according to the required specifications. In other studies, Zhang et al. [7] worked in MC using the effect of the model k-e turbulent in three types of cross-section for the MC. They used circular, trapezoidal, and rectangular sections, comparing the results obtained in terms of heat transfer and mechanics of the fluid, establishing the best design in terms of performance.

Naqiuddin et al. [8] conducted a study on the recent technological advances in electronic systems that use MC systems. In their study, they emphasized the importance of geometry, fluid mechanics, fluid type, phase change, and the applications of the same to improve heat dissipation. Kirsch and Thole [9] evaluated the behavior of MC developed by additive metal fabrication methods with simulated profiles through computational tools, comparing the effect on the performance of the device in the tests performed, highlighting the influence of the roughness present on the surfaces of the elements manufactured with respect to the change of the shape of the profiles.

Yue et al. [10] developed a study by CFD modeling of the segment of a steam turbine separated by MC varying the filling ratio of the refrigerant, finding the optimum operating percentages, and establishing a method to simplify the experiments. A more complete review of the main applications that use CFD for Physical phenomena modeling associated with heat transfer and fluid movement can be found in studies by Hnaien et al. [11] where they compared several turbulence models using computational mechanics to simulate the interaction of parallel jets, highlighting better predictions in the results when using the standard k - ε model.

Mohammed [12] reviewed literature focusing on the use of CM and nanofluids in both experimental and numerical designs, showing in his findings the improvement of heat transfer from devices because of increasing pressure drop. Mohd and Parashram N. [13] in their study used numerical techniques to predict the behavior of the dimensionless Nusselt number, through CFD simulations where the turbulence model SST Gamma-theta has greater precision in convergence. Aqilah et al. [14] performed a mesh effect study on CFD simulations of the aerodynamic profile in an airfoil, indicating the importance of mesh quality when comparing results with experimental studies. Drăgan [15] conducted a performance study of a centrifuge compressor using CFD software to simulate fluid behavior and heat transfer in equipment with optimized design under different operating conditions.

Thome [16] conducted a review study of the effect of boiling in MC devices, for CFD simulations and experiments with two-phase flows, and evaluated the behavior of heat transfer in MC when boiling. Talimi et al. [17] conducted a review on numerical simulation research in small channels in which the hydrodynamic and heat transfer characteristics in non-boiling gas-liquid and liquid-liquid slug flows were studied for circular and non-circular channels. Bagheri-Esfe and Manshadi [18] developed a code for CFD simulations estimating the behavior of flow fields to design a wind tunnel.

Azizi et al. [19] studied gas-solid heat transfer in risers using two-fluid CFD modeling to evaluate the effect of bed angle and particle feed velocity. Villegas et al. [20] used a coupling of computational tools, applying CFD to represent the friction-stir welding phenomenon. Culun et al. [21] developed a synergistic analysis in CFD of heat transfer of impinging multi jets, evaluating parameters such as the geometry of the jet holes, their arrangement, the density of the jets through spanwise and streamwise directions, and the type of confinement. Elsamni et al. [22] used CFD to study the characteristics of laminar flow in a semi-circular duct and the effect of the Reynolds number on the hydrodynamic development length and the friction factor, and subsequently compared them with straight and curved ducts.

Thus, computational modeling through CFD software of the design of the device with MC system starts from the advances of the computational technology of last generation processors, which require a heat removal system that satisfies the conditions for its performance over maximum efficiency. Traditional propeller heatsinks have been in the market for many decades obtaining a good performance in their operation, but as the technology of the processors increases, the system must meet their maximum operation demands. MC technology has had a great impact on cooling processes with good results. Based on these studies and technologies, the design and modeling of a device that satisfies the conditions of the last generation processor is generated.

In this study, a cooling system of an electronic device is evaluated, using water as the working fluid in a MC system, considering the heat transfer and the physical modeling involved in this case, in order to compare the operating temperature of the device with traditional forced airflow cooling. Finally, the safety factor of the MC design is verified in the numerical analysis.

2. MATERIALS AND METHOD

The development and application of phenomena associated with the behavior of CM technology are described by its formulation of mathematical models, as explained below, considering that the modeling equations were made with the use of ANSYS® software.

It begins with a description of the physical phenomena that govern the system. Similarly, the boundary conditions for heat conduction and the thermodynamic relationship of the working fluid with convection heat transfer are exposed. The external and internal working fluids were studied through the logarithmic mean temperature difference to establish the change of temperature in the exchange device system. From the description of the heat transfer model, the variables involved in the process are established to use a computational tool where these values will be entered. Previously, a CAD (Computer-Aided Design) model was developed. Then, an ANSYS CFX tool was used to evaluate the energy, continuity, moment, and turbulence models to obtain the results.

2.1 Physical Modeling

The heat transfer phenomenon to which the MC cooling device is subjected, together with the working fluid and heat dissipation in the computer processor, requires a mathematical model to describe its behavior. Therefore, the following numerical formulations are established. Heat input from processor to MC device is:

$$\frac{\dot{Q}_{CPU}}{A_{s,CPU}} = -k_d \frac{\partial T(0,t)}{\partial y} \tag{1}$$

where k_d is the thermal conductivity of the solid material with which the device MC was designed, $\dot{Q}_{CPU}$ is the heat generated by the processor, $A_{s,CPU}$ is the surface area of contact between the processor and the device MC. Consequently, the liquid used for cooling inside the MC is water, and the heat transfer rate of the coolant without phase change is described with equations (2-3):

$$\dot{Q}_{H2O} = C_{pH2O}\dot{m}_{H2O}(T_{out,H2O} - T_{in,H2O}) = h_{mc}A_{s,mc}(\Delta T_{\ln,mc}) \tag{2}$$

$$\Delta T_{\ln,mc} = \left(\frac{T_{in,H2O} - T_{out,H2O}}{\ln\left(\frac{T_{s,prom} - T_{out,H2O}}{T_{s,prom} - T_{in,H2O}} \right)} \right) \tag{3}$$

Assuming in the first stage of calculation the surface temperature of the MC as average temperature. Where $\dot{m}_{H2O}$ is the mass flow of the coolant (water) and C_{pH2O} is the specific heat, $T_{in,H2O}$ and $T_{out,H2O}$ are the coolant inlet and outlet temperatures in the MC, $T_{s,prom}$ is the average temperature on the surface of the MC, $\Delta T_{\ln,mc}$ is the average log temperature of the MC, h_{mc} is the coefficient of heat transfer by internal convection in the MC. For obtaining the coefficient of heat transfer by convection the number of Nusselt Nu is calculated by Dittus-Boelter equation (4).

For the turbulent regime:

$$Nu = \frac{h_{mc}D_d}{k_{H2O}} = 0.023\,\mathrm{Re}_D^{0.8}\,\mathrm{Pr}^n \tag{4}$$

For the laminar regime with constant heat flux, and the laminar regime with an average surface temperature of constant MC, given the dimensions of the MC, the Nusselt number is:

$$Nu_{\dot{q}} = 4.12; \quad Nu_{Ts} = 3{,}308; \quad f = 61.144 / \mathrm{Re}_D \tag{5}$$

where k_{H2O} is the thermal conductivity of the coolant liquid (water), D_d is the hydraulic diameter before the test section at the entrance, f is the friction factor and the Reynolds number represented by D_d is calculated according to equations (6-7):

$$\mathrm{Re} = \frac{\vec{V}D_d}{v} \tag{6}$$

$$D_d = \frac{4A_c}{p} \tag{7}$$

https://doi.org/10.31436/iiumej.v23i1.2113

where $\vec{V}$ represents the velocity of the coolant (water), v is the kinematic viscosity of the coolant (water), A_c is the area of the cross-section located before the test zone and p is the perimeter of this.

The heat generated around the MC device is represented by the equation (8):

$$-k_d \frac{\partial T(L,t)}{\partial y} = h_{2,\infty}\left[T(L,t) - T\infty\right] + \varepsilon\sigma\left[T(L,t)^4 - T\infty^4\right] \quad (8)$$

where $h_{2,\infty}$ is the coefficient of free heat transfer of the material of the device MC with the air environment of the surroundings, $T\infty$ is the environment air temperature of the surroundings, σ is the Stefan-Boltzmann coefficient for radiation, ε is the emissivity of the surface of the MC device. Figure 1 describes the system developed.

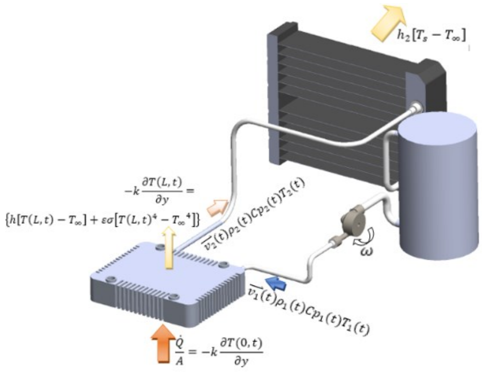

Fig. 1: Representation of the cooling system with the use of MC.

2.2 Computational Model

The design of the MC cooling device was generated because of the size specifications of the last generation processor studied. That is, a compact device made of commercial aluminum that consists of an internal fluid inlet of 3.8 mm in diameter, the constitution of the series of MC of height 0.9 mm and width 0.5 mm, a fluid outlet of 3.8 mm in diameter, as shown in Fig. 2.

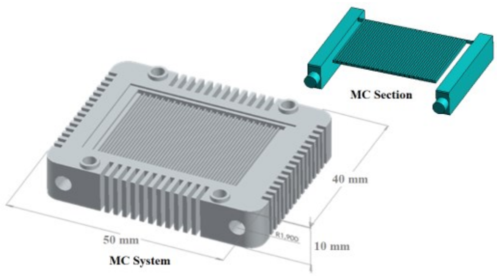

Fig. 2: Geometric representation of the cooling device by MC.

Figure 3 shows the volumetric meshing of the system in which the size of the elements is stipulated for a minimum of 0.015 mm and a maximum of 0.3 mm with a growth rate of 1.2, obtaining an element quality of 82 % indicating a good mesh, for a ratio of 369,995 nodes and 1,047,186 elements in the microchannel system. These meshing data are selected by performing different meshing independence tests according to a response variable, obtaining an error percentage of 0.6%.

The input boundary condition is assigned by the velocity variables and the ambient temperature of 298.15 K of the water domain, the velocity is calculated by the flow and the transverse area of the input.

The boundary condition of the MC cooling device domain presents a set of variables in the initial state, the initial temperature of the medium at 298.15 K, free convection of the medium with the aluminum material at 25 W/m^{2}K. The processor generates a heat flux corresponding to its maximum operation consumption on the surface or base of the MC cooling device and this corresponds to 22,849 W/m^2 by equation (1).

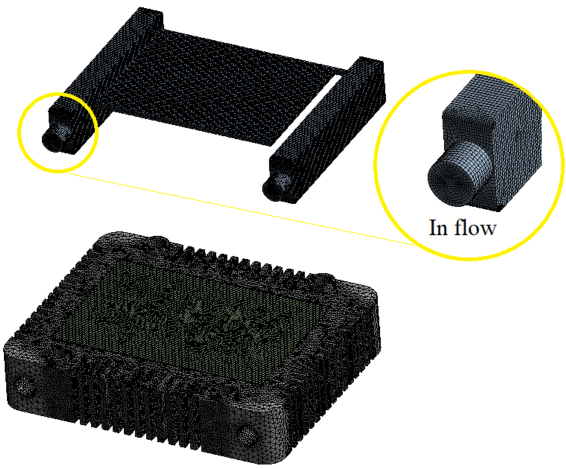

Fig. 3: Volumetric mesh of the microchannel section.

Therefore, the models applied to the system consist of the movement of the working fluid and the heat transfer between the contact interfaces and the environment, for which, the computational models of momentum conservation, energy conservation, and turbulence models for fluid movement are shown in equations (9) to (12).

Momentum conservation:

$$\frac{\partial}{\partial t}(\rho\vec{v})+\nabla(\rho\vec{v}\vec{v})=-\nabla p+\nabla\left[\mu\left(\nabla\vec{v}+\nabla\vec{v}^T\right)\right]+\rho\vec{g}+\vec{F} \tag{9}$$

where ρ is the density of the fluid, μ is the dynamic viscosity, p is the pressure, is the dynamic viscosity of the fluid, $\vec{v}$ is the velocity of the fluid.

$$\frac{\partial}{\partial t}(\rho\vec{v})+\nabla(\rho E)+\nabla\vec{v}(\rho E+p)=\nabla p\left(k_{eff}\nabla T\right)+S_h \tag{10}$$

The equation (10) represents the energy conservation of the system, where the variable k_{eff} is the coefficient of effective thermal conductivity, and S_h represents the energy of the potential chemical system.

Turbulence model *k-ε*:

$$\frac{\partial}{\partial t}(\rho k)+\frac{\partial}{\partial x_i}\left(\rho k u_j\right)=\frac{\partial}{\partial x_i}\left[\left(\mu+\frac{\mu_i}{\partial_k}\right)\frac{\partial k}{\partial x_i}\right]+G_k+G_b-\rho\varepsilon-Y_M+S_k \tag{11}$$

$$\frac{\partial}{\partial t}(\rho\varepsilon)+\frac{\partial}{\partial x_j}\left(\rho k u_j\right)=\frac{\partial}{\partial x_j}\left[\left(\mu+\frac{\mu_i}{\partial_z}\right)\frac{\partial s}{\partial_j}\right]+\rho C_1 S_\varepsilon-\rho C_2\frac{\varepsilon^2}{k+\sqrt{v\varepsilon}}+C_{1z}\frac{\varepsilon}{k}C_{3z}G_b+S_z \tag{12}$$

https://doi.org/10.31436/iiumej.v23i1.2113

where G_k represents the turbulence generation of the kinetic energy due to the average velocity gradient, Y_M represents the contribution of the incompressible turbulence of fluctuating expansion to the general dissipation rate, and G_b is the generation of turbulence due to the floating kinetic energy. Due to defined boundary conditions and interface domains, a convergence criterion of 1E-04 is established to start the CFD software solver. Table 1 presents the boundary conditions applied to the computational model with the process conditions.

Table 1: Boundary conditions

Conditions	Value
Initial temperature	298.15 K
Heat Flux	22.849 kW/m^2
Pump's rotational Speed	100 – 2000 rpm
External heat transfer coefficient of the microchannel	25 W/m^{2}K
External environment Temperature	298.15 K
Inlet and outlet pipe area	11.341 mm^2
The cross-sectional area of the microchannel	**1.45**m^2

3. RESULTS AND DISCUSSION

The transfer models that govern the design of the MC system were indicated in the modeling of the device. The design was generated from two aluminum materials for the MC device with water as the cooling liquid. The MC cooling device was generated based on the size specifications of the studied computer processor, that is, a compact device using MC in series, the boundary conditions assigned by the variables of velocity and ambient temperature of the water domain, and the limit conditions of the MC cooling device presenting a set of variables in an initial state such as ambient temperature. After the CFD software solution process was complete, the temperature, pressure, and velocity profiles were rendered for analysis purposes. The reading of results was presented in two specific areas, the interior of the MC and the contact surface of the processor-cooling device in planes, therefore, the results of the study carried out are shown in Fig. 4 and 5.

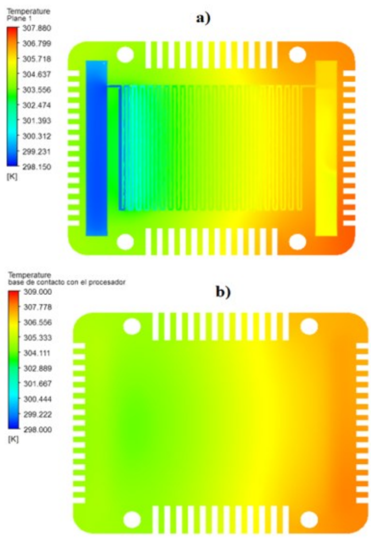

Fig. 4: Temperatures profiles: a) Region of MC, b) Region of processor-cooling device contact.

https://doi.org/10.31436/iiumej.v23i1.2113

Figure 4 shows the distribution by contours of the temperature in the device and inside the MC, from which it can be noted that the coolant reached a working temperature of approximately 306 K, while the solid-body reached a temperature of 307.5 K and a minimum temperature of 304 K located in the inlet zone of the coolant to the system. These results of MC temperature profiles were compared with those proposed by Naqiuddin et al. [6], Moradikazerouni et al. [23], and Abdollahi et al. [24]. It was observed that the temperature differences were greater by up to 5 K according to the simulation conditions demonstrating that the operating principle of liquid cooling is adequate to maintain the working conditions of the electronic element, meaning the processor.

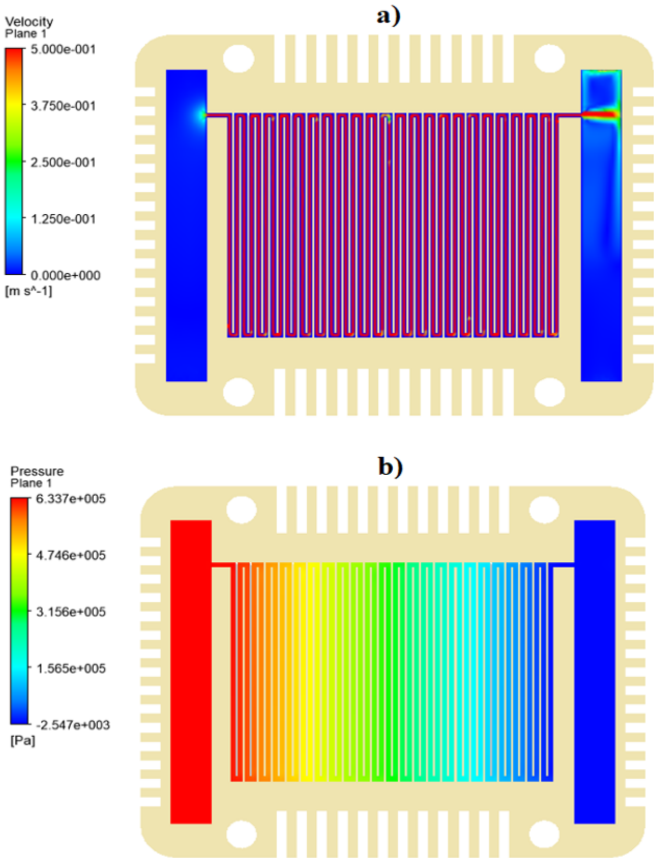

Fig. 5: Profiles in the MC region: a) Speed of the cooling fluid, b) Pressure in the cooling fluid.

In Fig. 5, the velocity and pressure profiles corresponding to the working fluid inside the MC are observed, from which it can be seen that for the flow rate established when using equation (9), the operating pressure at the beginning of the conduit is 633.7 kPa and the speed reached by the liquid in the MC is 0.5 m/s. This relationship shows that increasing the fluid velocities requires a greater pressure range so that the flow parameters remain at the working condition that is required by the processor in terms of temperature. This velocity result is within the range of results obtained by Chiu et al. [3] on hydraulic simulations in the CFDs of microchannels used as heatsinks.

When graphing, there is a line along the length of the device, these represent the temperature curves in two regions, which appear as a result of the system temperature is stabilized from 299 K to 306.4 K together with the MC and the temperature in the region of contact of the device with the processor from 304.5 K to 307.4 K, in the range of the optimum operating temperature of the processor as seen in Fig. 6.

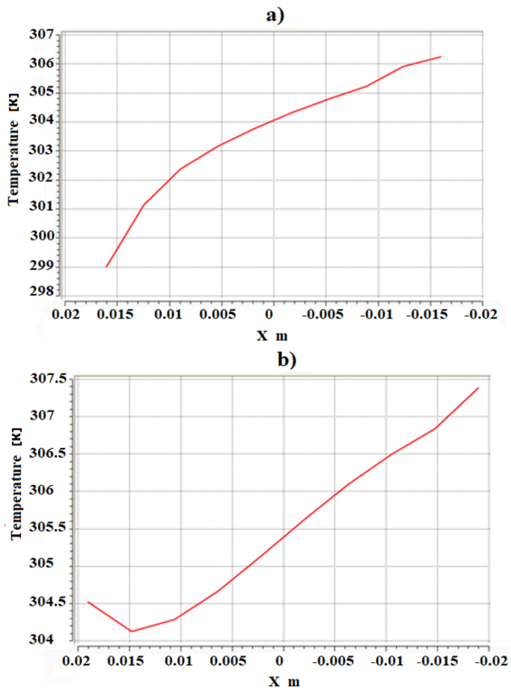

Fig. 6: Temperature curves: a) Region of MC, b) Region of contact processor-cooling device.

The analysis of the behavior of the stabilized temperature in the computer processor was carried out, varying the range of angular velocity from the water pumping system to the MC device, thus obtaining characteristic curves that allowed choosing the best range of speeds to model. Figure 7 shows the graphs of the temperature profile reached in the computer processor, for each point of variation of the angular velocity in the pumping system. It is observed that the angular velocity ranges between 500 rpm and 1000 rpm do not represent a significant variation in the temperature obtained in the computer processor, so this range of velocity is a good option to model the process.

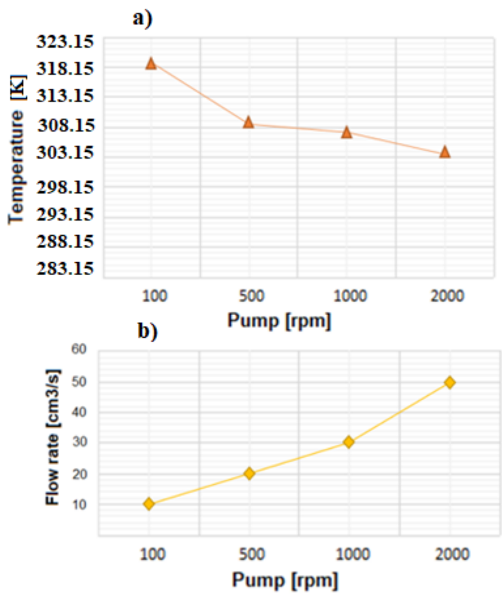

Fig. 7: Temperature & flow rates curves: a) Region of Processor, b) Region of contact processor-cooling device.

Once the fluid dynamic behavior of the working liquid inside the MC system has been evaluated, a multiphysics coupling is made in the simulation software, by importing the pressures reached inside the MC system by the working fluid, in order to analyze in a structural mechanics tool, the stresses caused by this pressure on the walls of the MC. Figure 8 shows the pressure vectors transferred to the structural mechanics' tool.

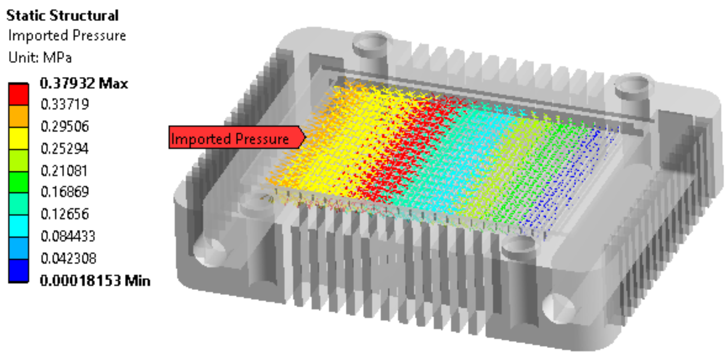

Fig. 8: Pressure imported from CFD tool to structural mechanics' tool.

Upon obtaining the results of the temperature and pressure profiles the coupling between computational tools was performed to determine the effect caused by the pressures reached by the working fluid and the stress it causes on the MC surfaces. The magnitude of the stresses applied to the MC surfaces was obtained using commercial aluminum as the device material, in addition to estimating the safety factor to evaluate the feasibility of manufacturing a physical design under these working conditions. These results are shown in Fig. 9.

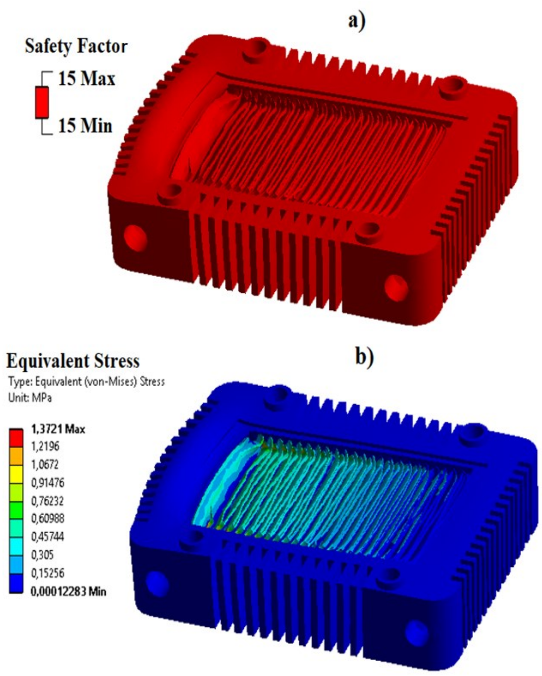

Fig. 9: Design contours: a) Security factor, b) Von Mises stress.

Figure 9 can be analyzed to reveal the maximum applied stress for the pressure conditions generated for the cooling of the electronic processor device. The maximum applied stress is 1.38 MPa with a safety factor of 15 granting design reliability for the use of a Wide field of pressures used for cooling electronic devices.

Sreehari and Sharma [25] carried out research about the thermal performance in MC using water as the working fluid. The effect of three types of surface engraving that were modeled were compared (rectangular coil, U-curvature coil, and V-shaped coil) and subjected to computational and experimental study. These microchannel systems have a rectangular cross-section with dimensions of 0.6 mm x 0.3 mm using heat flux of 10 kW/m^2, 20 kW/m^2, and 30 kW/m^2. The results have shown an acceptable deviation between the experimental and numerical results with a maximum value of 11%. The rectangular coil behaved similarly to the device analyzed in the present investigation.

4. CONCLUSION

The main motivation for carrying out this study was the fact that the usual working temperatures of a CPU range between 328.15 K and 343.15 K when using the traditional dissipation system for air ventilation, which affects the performance of the device, causing forced labor. On this basis, the following conclusions are drawn when liquid cooling is used in a microchannel-type device.

The main result is that the variables analyzed in the study such as fin geometry and MC distribution influence the heat transfer less than the flow rate used. For low flow rates, the temperature tends to exceed the normal operating range of the system, which implies that the design of the device must have the corresponding mechanical properties for the pressure range in the MC, as well as the supply hoses and the pumping device.

The results of the simulation of the system were analyzed and it was observed that the behavior of the MC cooling device, when the processor is at its maximum use in operations, maintains an optimum working temperature in an average range of 307 K. It should be noted that the value depends on the conditions of fluid entry, such as the flow rate.

It is concluded that at a lower flow rate than the one used in the fluid, the behavior of the temperature exceeds the range of 313 K, which is still suitable for optimal operation for the studied processor which establishes a limit of operation at 333 K and critical state at more than 373 K. The design of the device is suitable for its operation since it has a safety factor of 15 for optimal process conditions using commercial aluminum material, which is a light material and good conductor.

The mesh size was evaluated with respect to the temperature reached by the contact surface with the computer processor. varying the number of elements in the mesh from a default value of 320,775 to 1,047,186, obtaining an error percentage between samples of 0.6%, which is an acceptable value for the reliability of the simulation.

This study consisted of analysis using computational tools to simulate the explained phenomenon. Validations were carried out with works published by various authors applying similar simulation techniques. The experimental validation of the design will be carried out in future studies.

REFERENCES

[1] Belhardj S, Mimouni S, Saidane A, Benzohra M. (2003) Using microchannels to cool microprocessors: A transmission-line-matrix study. Microelectronics Journal, 34(4): 247–253. https://doi.org/10.1016/S0026-2692(03)00004-1

[2] Chen CH, Ding CY. (2011) Study on the thermal behavior and cooling performance of a nanofluid-cooled microchannel heat sink. International Journal of Thermal Sciences, 50(3): 378–384. https://doi.org/10.1016/j.ijthermalsci.2010.04.020
[3] Chiu HC, Jang JH, Yeh HW, Wu MS. (2011) The heat transfer characteristics of liquid cooling heatsink containing microchannels. International Journal of Heat and Mass Transfer, 54(1–3): 34–42. https://doi.org/10.1016/j.ijheatmasstransfer.2010.09.066
[4] Bosi F, Balestri G, Ceccanti M, Mammini P, Massa M, Petragnani G, Ragonesi A, Soldani A. (2011) Light prototype support using micro-channel technology as high efficiency system for silicon pixel detector cooling. Nuclear Instruments and Methods in Physics Research, Section A: Accelerators, Spectrometers, Detectors and Associated Equipment, 650(1): 213–217. https://doi.org/10.1016/j.nima.2010.12.187
[5] Brinda R, Joseph Daniel R, Sumangala K. (2012) Ladder shape micro channels employed high performance micro cooling system for ULSI. International Journal of Heat and Mass Transfer, 55(13–14):3 400–3411. https://doi.org/10.1016/j.ijheatmasstransfer.2012.03.044
[6] Naqiuddin NH, Saw LH, Yew MC, Yusof F, Poon HM, Cai Z, Thiam HS. (2018) Numerical investigation for optimizing segmented micro-channel heat sink by Taguchi-Grey method. Applied Energy, 222: 437–450. https://doi.org/10.1016/j.apenergy.2018.03.186
[7] Zhang Y, Wang S, Ding P. (2017) Effects of channel shape on the cooling performance of hybrid micro-channel and slot-jet module. International Journal of Heat and Mass Transfer, 113: 295–309. https://doi.org/10.1016/j.ijheatmasstransfer.2017.05.092
[8] Naqiuddin NH, Saw LH, Yew MC, Yusof F, Ng TC, Yew MK. (2018) Overview of micro-channel design for high heat flux application. Renewable and Sustainable Energy Reviews, 82: 901–914. https://doi.org/10.1016/j.rser.2017.09.110
[9] Kirsch KL, Thole KA. (2018) Isolating the effects of surface roughness versus wall shape in numerically optimized, additively manufactured micro cooling channels. Experimental Thermal and Fluid Science, 98: 227–238. https://doi.org/10.1016/j.expthermflusci.2018.05.030
[10] Yue C, Zhang Q, Zhai Z, Ling L. (2018) CFD simulation on the heat transfer and flow characteristics of a microchannel separate heat pipe under different filling ratios. Applied Thermal Engineering, 139: 25–34. https://doi.org/10.1016/j.applthermaleng.2018.01.011
[11] Hnaien N, Marzouk KS, Ben HA, Jayb J. (2016) Numerical Study of Interaction of Two Plane Parallel Jets. International Journal of Engineering, 29(10): 1421–1430. https://doi.org/10.5829/idosi.ije.2016.29.10a.13
[12] Mohammed HA, Bhaskaran G, Shuaib NH, Saidur R. (2011) Heat transfer and fluid flow characteristics in microchannels heat exchanger using nanofluids: A review. Renewable and Sustainable Energy Reviews, 15(3): 1502–1512. https://doi.org/10.1016/j.rser.2010.11.031
[13] Mohd Umair S, Parashram Gulhane N. (2016) On numerical investigation of non-dimensional constant representing the occurrence of secondary peaks in the Nusselt distribution curves. International Journal of Engineering, Transactions A: Basics, 29(10):1 431–1440. https://doi.org/10.5829/idosi.ije.2016.29.10a.00
[14] Aqilah F, Islam M, Juretic F, Guerrero J, Wood D, Nasir Ani F. (2018) Study of mesh quality improvement for. IIUM Engineering Journal, 19(2): 203–212. https://doi.org/https://doi.org/10.31436/iiumej.v19i2.905
[15] Drăgan V. (2017) Centrifugal compressor efficiency calculation with heat transfer. IIUM Engineering Journal, 18(2): 225–237.
[16] Thome JR. (2004) Boiling in microchannels: A review of experiment and theory. International Journal of Heat and Fluid Flow, 25(2): 128–139. https://doi.org/10.1016/j.ijheatfluidflow.2003.11.005
[17] Talimi V, Muzychka YS, Kocabiyik S. (2012) A review on numerical studies of slug flow hydrodynamics and heat transfer in microtubes and microchannels. International Journal of Multiphase Flow, 39: 88–104. https://doi.org/10.1016/j.ijmultiphaseflow.2011.10.005
[18] Bagheri-Esfe H, Manshadi MD. (2018) A low-cost numerical simulation of a supersonic wind-tunnel design. International Journal of Engineering, Transactions A: Basics, 31(1): 128–135. https://doi.org/10.5829/ije.2018.31.01a.18
[19] Azizi K, Keshavarz Moraveji M. (2017) Computational fluid dynamic- two fluid model study of gas-solid heat transfer in a riser with various inclination angles. International Journal of

Engineering, Transactions A: Basics, 30(4): 464–472. https://doi.org/10.5829/idosi.ije.2017.30.04a.02

[20] Villegas JF, Guarín AM, Unfried-Silgado J. (2019) A Coupled Rigid-viscoplastic Numerical Modeling for Evaluating Effects of Shoulder Geometry on Friction Stir-welded Aluminum Alloys. International Journal of Engineering, Transactions B: Applications, 32(2): 184–191. https://doi.org/10.5829/ije.2019.32.02b.17

[21] Culun P, Celik N, Pihtili K. (2018) Effects of design parameters on a multi jet impinging heat transfer. Alexandria Engineering Journal, 57(4): 4255–4266. https://doi.org/10.1016/j.aej.2018.01.022

[22] Elsamni OA, Abbasy AA, El-Masry OA. (2019) Developing laminar flow in curved semi-circular ducts. Alexandria Engineering Journal, 58(1): 1–8. https://doi.org/10.1016/j.aej.2018.03.013

[23] Moradikazerouni A, Afrand M, Alsarraf J, Mahian O, Wongwises S, Tran MD. (2019) Comparison of the effect of five different entrance channel shapes of a micro-channel heat sink in forced convection with application to cooling a supercomputer circuit board. Applied Thermal Engineering, 150: 1078–1089. https://doi.org/10.1016/j.applthermaleng.2019.01.051

[24] Abdollahi A, Mohammed HA, Vanaki SM, Osia A, Golbahar Haghighi MR. (2017) Fluid flow and heat transfer of nanofluids in microchannel heat sink with V-type inlet/outlet arrangement. Alexandria Engineering Journal, 56(1): 161–170. https://doi.org/10.1016/j.aej.2016.09.019

[25] Sreehari D, Sharma AK. (2019) On thermal performance of serpentine silicon microchannels. International Journal of Thermal Sciences, 146: 1-14. https://doi.org/10.1016/j.ijthermalsci.2019.106067