Heat transfer improved and turbulent hydrodynamic characteristics: pin-fin heat sinks

N. Bessanane[1,2], M. Siameur[1], M. Rebay[2] and Y. Demagh[1]

[1]*LESEI Laboratory/Faculty of Technology, University of Batna 2,1Av. Chahid Boukhlouf Med Elhadi, 05001 Batna, Algeria, n.bessanane@univ-batna2.dz*

[2]*ITHEMM Laboratory, University of Reims Champagne-Ardenne, Faculty of Sciences PB1039, Reims 51687, France.*

Abstract – Pin-fins are frequently used to improve the heat transfer surface and promote turbulent motion, which improves the device's cooling process by enhancing heat dissipation, as in electronic devices and different cooling systems for industrial applications. The application has a burst out this last decade and became vital in several industrial devices. The present study is a numerical investigation of flow and heat transfer in pin-fin heatsinks (PFHS). The pin-fins have a diamond shape arranged in a segregated disposition (corrugated channel). To adequately calculate the heat transfer coefficient within this complex thermal system; several parameters, such as mass flow rate, geometry dimensions, heat flux and reference temperature are extensively examined. The importance in the way the correct estimation of the heat transfer coefficient led to better optimization of the cooling process performances. This work aimed to elaborate a parametric study to correctly estimate the temperature difference between the cooler fluid and the heat sink wall. For this purpose, a comparative method of 3-D stationary numerical simulations was conducted between laminar and different turbulence models (standard k–ε, RNG k–ε, Realizable k–ε, standard and SST k–ω), and under turbulent conditions allowing us to compare the characteristic flow effects. We are interested mainly in the determination of the better approach for heat transfer coefficient estimation. An approach with variable reference temperature (VRT) has been adopted in the calculation of the wall-fluid temperature difference. The numerical procedure has been validated by experimental measurements. The proposed methodology to calculate the reference temperature leads to a better presentation of the heat transfer coefficient, in particular the variation of the averaged heat transfer coefficient against Reynolds number. The results obtained show that the model Realizable k–ε is better because it gives more precise results, which are from the physical point of view and closer to the experimental one.

Keywords: micro/mini-channels, pin-fin heat sinks, heat transfer enhancement, turbulence, Re effects, CFD.

1. Introduction

The extended heat transfer surface is a key parameter in cooling systems. Its improvement retains continually great attention in engineering applications. It has been an outburst this last decade and become fundamental in several industrial devices, as a passive technique commonly taken on for enhancing heat transfer. The cooler is pure demineralized water and its thermophysical properties are considered to vary with temperature according to the polynomial law. Pin fins are frequently used for this purpose to promote especially the cooling process and the device's durability. Therefore, various types of pin fins are regularly used for both natural and forced convection heat transfer. In this context, several geometrical (shape, arrangement, dimensions...) and hydrodynamic parameters are extensively examined in many research works.[1-3]

Diamond shaped pin-fins are often used in cooling devices to enhance heat dissipation by

increasing surfaces and promoting turbulence activity. The pin-fins geometrical arrangement and dimensions factors are extensively examined in many research works, as well as thermal-fluid properties, such as Zukauskas and Moores [4,5], Shkarah et al. [6].
Mini/micro channels are regularly used for the intensification of heat dissipation in miniature cooling devices by increasing the interface area of solid/fluid (contact region). Various forms of fins and channels are typically employed for both laminar and turbulent flow regimes. The geometrical dimension factors are extensively examined in many research works, as well as thermal-fluid properties. for example, the hydraulic diameter (Dh) of the micro-channels are considered between 10 and 1000µm (Kandlikar et al. [2004], Kosar [2006], Morini [2004] and Rebay et al. |2016]). [7,8,9,10]

The effects of the evaluation of the Reynolds number on thermal and hydrodynamic quantities for this kind of geometry (small scales) is not enough documented in the literature. This is fact is due to the lack of details about the reference quantities; velocity and hydraulic diameter for the evaluation of Re. Furthermore, this dimensionless number becomes more sensible to be defined in terms of small scale and complex configurations, leading to significant effects on thermal and hydrodynamic quantities. In other terms, computations of pressure drop, friction coefficient, Nusselt number, convective coefficient, thermal resistance with the consideration of the reference velocity (inflow, averaged or maximum velocity), can lead to discrepancies results. The present study aims then to unification of this divergence point, as majority of previous studied cases, unnoticed more precision on *Re* evaluation. Two approach are used in the open literature. The first approach is based on inflow velocity as reference value. Indicatively, the recent study of Ahmadian [11] regarding the parametric optimization of the pin-fin heat sink to improve its thermal and hydraulic characteristics. However, the calculation of Re is not specified. Begag et al. [12] presented a numerical analysis of the thermos-convective behavior of a turbulent flow in a mini-channel in the presence of V-shape ripples, they used Re based on the inflow velocity as average quantity. Similarly, Charef-Khodja et al. [13] presented a numerical study of the flow in a rectangular micro channel, the inflow velocity was used for Re calculation. In revenge, the work of Al-Haddad [14] related to mini-channel, the average flow velocity was considered. Furthermore, Debray [15] has conducted an experimental work on measurement of heat transfer coefficients by forced convection in mini-channels. The Reynolds number is based on the average velocity of the involved flow rate, for both cases of laminar and turbulent regime. The authors noticed that *Re* estimation augmented noticeably (twice compared to the first evaluation approach) for channel's fluid passage value of about 1000µm (Morini [16]).

This variation is related to the effect of viscosity (encounter of boundary layers of opposite walls), followed by a decrease of friction coefficient. However, the deviation observed by the authors in the laminar regime ($Re \leq 400$) is not completely explained, namely the increase of Nu with Re, indicating the presence of a flow transition regime located between $Re=200$ and $Re=400$. In recent research work, B.N. et al. [17-19] presented different approaches for the calculation of the Reynolds number, depending on several choice of the reference velocity with the appropriate hydraulic diameter, especially when dealing with non-regular configurations (complex geometries).

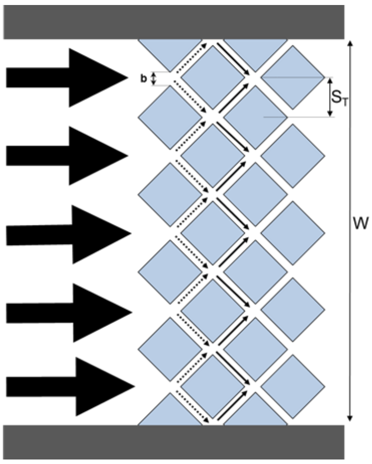
Concerning the evaluation of the heat transfer coefficient, several works used the arithmetic mean of the two inlet and outlet temperatures as the reference temperature of the fluid. Yang [20] proposed the use of ($\Delta Tm = Tw - Tbulk$) as temperature difference to evaluate heat transfer coefficient. Jiang [21] carried out experimental and numerical studies about micro heat sinks, the reference temperature was based on the "bulk" temperature of the fluid throughout the channel volume. However, in this present study the reference temperature is described in B.N.

et al. [19].

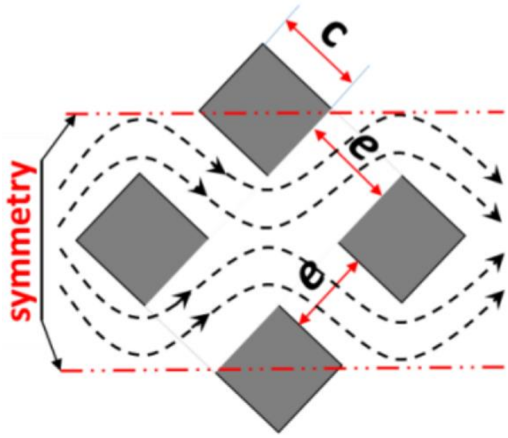
In this work, a series of tests was carried out on two configurations, one of which was similar to that studied experimentally by Rebay et al [10]. Different turbulence models were compared (Sk-ε, RNGk-ε, Rk-ε, SSTk-w, RSM and LES) for flow rates of (0.3 - 3 *kg/min*). and some fluid flow cross-sections, with fluid passage width (*0.35mm-1.2mm*). The flow regime is considered stationary for all turbulence models, with the exception of the LES model, which is unsteady in nature. A confrontation of turbulence models was carried out to select the most appropriate for numerical modeling of this type of geometry. A better representation of the hydrodynamic and thermal field is obtained by the "Realisable k-ε" model.

2. Problem description

The mini-channel heat sinks considered in this study are used in the cooling process of the Synchrotron SOLEIL components (absorbers, mirrors, monochromators…etc.). SOLEIL, an acronym for "Optimized Source of Intermediate Energy Light's of LURE (Laboratory for the Use of Electromagnetic Radiation)" the research center in Paris (France), a particle (electron) accelerator that produces synchrotron radiation, an extremely powerful source of light that permits exploration of inert or living matter.



(a)



(b)

Figure 1: Studied configuration : (a) the considered mini-channels scheme (b) characteristic dimensions of pin-fin heat sink

The staggered pin fins arrangement has been assumed in our study as it is well known in literature to perform a better more heat transfer improvement [20-22].

The heat sink shape is a parallelepiped (Figure 1). The heat sink length is $L=100mm$ and its width and height are respectively $w=16$ *mm* and $H=1.6$ *mm*. All fins have the same height of channel *(H)* and a fin square base dimensions $C\times C=1.7\times1.7mm^2$ (Figure 2). The top and the bottom of the domain are planes with *3mm* thickness. The distributor length (*inlet*) is about *5mm* and the collector length (*output*) is equal to *12mm*; the origin plot is fixed at *5mm* and the downstream evolution is ends at *105mm*. the fluid passage spacing values are (*e=0.35–0.7–1.2 and 1.6 mm*).

In this study, the geometry of the reference dissipater of the type mini-channels is diamond shaped pin-fins, with the same bulk volume. Different geometries with different fluid passage dimensions (e) were considered; only two cases are presented, which are carried out at different flow rates between (*0.05 and 3 kg/min*).

3. Mathematical formulation and numerical approach

3.1. Governing equations

The CFD-code FLUENT offers different approaches for treating turbulent flows in heat dissipater and cooling systems. The equations used by the finite volume solver to model the 3D steady state flow are the incompressible, continuity, Reynolds-averaged momentum and energy equations. The simple algorithm was used for pressure–velocity coupling. The second order "Up-wind" scheme is chosen for discretization of the governing equations.
In our study, we have involved the available RANS-type models: as we wish to obtain a good approximation of mean values in steady-state flows. The main models used are the two families of k-ε and k-ω. We will restrict ourselves to those models that reflect the wall treatment approach. The standard k-ω, SST and Spalart-Allmaras models, which apply in boundary layers if the mesh resolution is sufficient (for conventional scales), have therefore been discarded.
We have tested the different turbulence models for different mass flow rates, in order to select the model that best accounts for the turbulence generated. In order to save computation time, we first studied the dynamic problem only.

3.2. Boundary conditions

The boundary conditions can be summarized as follows in Figure 2 and detailed in table 1:

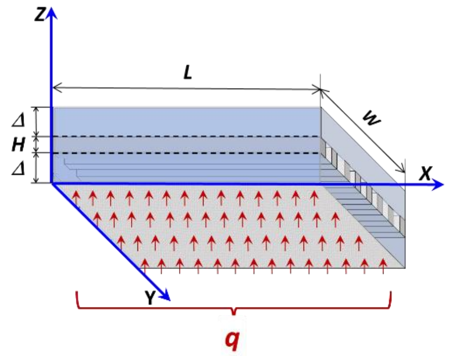


Figure 2: Characteristics of the studied geometry

Table 1: Boundary condition.

Position	Type of Boundary
z=0	: uniform heat flux *q* at the base plan (x,y)
z=Δ ; Δ+H	: fluid-solid conjugate heat transfer conditions
z=2Δ+H	: adiabatic on top plan (x,y)
y=0 ; y=W	: adiabatic on lateral plan (x,z)
x=0 ; x=L	: adiabatic on plan (y,z)
x=0	: inlet velocity and fixed T=303°C
x=L	: outflow conditions

A symmetry plans are considered with no heat exchange and conjugate heat transfer conditions were considered at the solid-fluid interface:

3.3 Numerical procedures

The Reynolds number in the channel fluid passage was treated specifically with great attention to estimate it with different approaches. These are special approaches to obtain Re_{max} and Re_{ave} based on the hydraulic diameter of a channel corresponding to the bulk volume of fluid channel (figure 1-a). These approaches follow exactly the flow path between fins (channels) when the control surface taken is $(e\times H)$, perpendicular to the flow direction (figure 1). The maximum velocity is calculated at the smallest cross section of the fluid passage, inside the channel embedded with pin-fins array.

$$\mathrm{Re}_{max} = \frac{U_{\max} D_h}{\nu}; \mathrm{Re}_{ave} = \frac{U_{ave} D_h}{\nu} \text{ and } D_h = \frac{2eH}{e+H}$$

The new approach to compute the heat transfer coefficient elaborated in this research is as follows:

$$F_{bulk} = \frac{1}{4}\left(F_{bulk}\big|_{top} + F_{bulk}\big|_{bottom} + F_{bulk}\big|_{left} + F_{bulk}\big|_{right}\right)$$

Where F replaces the average quantities: q_w, T_w, h_{bulk}, Nu_{bulk} or f_{bulk} at the interfaces of the mini-channel. Bulk quantities for the interface walls are calculated according to bulk values of the four channel interfaces (top, bottom, left and right).

4. Results and discussion

A comparison is made by comparing experimental data, presented in figure 5, for the type of pin-fin heatsink, with a contracted fluid passage (*350µm*) and hydraulic diameter of 0.*574mm* respectively (*Dh*<1mm), where flow is considered turbulent even at low Reynolds numbers (Re<1000). A transition region of the flow regime (laminar-turbulent) is detected in the neighborhoods of Reynolds number values 200 to 300 (figure 6).
The calculation of the Reynolds number is defined according to the methods used by the authors [10,15,20], i.e. depending on how the reference velocity and hydraulic diameter are considered.

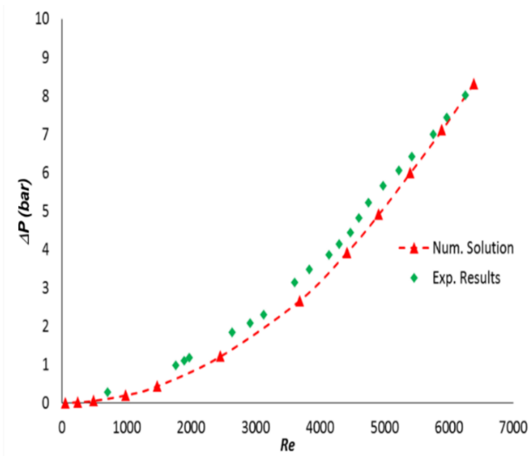


Figure 5: Validation of results for pin-fin heat sink *(e = 0.35 mm)*

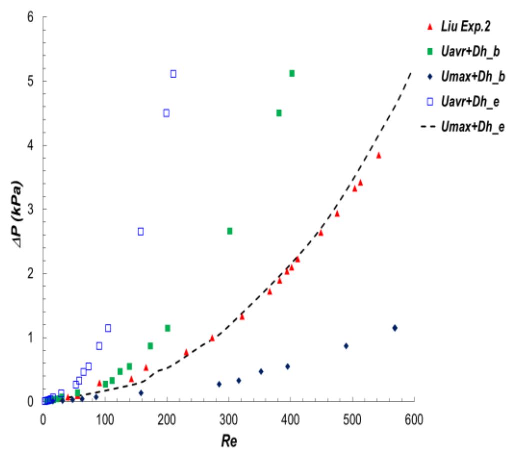


Figure 5: Pressure drop vs Re *(e=0.355mm) [20]*

Numerical simulations were carried out for the configurations of the diamond shaped pin-fin heat sink (with corrugated mini-channel PFHS); corresponding to four different fluid passages are discussed, where fluid flow rate values vary for different values of Reynolds number. The results are presented corresponding to different mass flow rates and different geometry dimensions.

In Figure 5, we can see a rapprochement between the two models *R k-ε* and *RNG k-ε,* with a gap before the regime is established. The rest of the models exhibit remarkable disagreement.
It can be seen that with increasing flow rate (figures 3,4 and 5), a slight difference is recorded between the three k-ε models (Std, "Realesable" and RNG), while for the two models Std k-ω and SST k-ω the gap remains significant. As a result, discrepancies were recorded for the laminar model when calculating with a slightly higher flow rate (Re<1200).

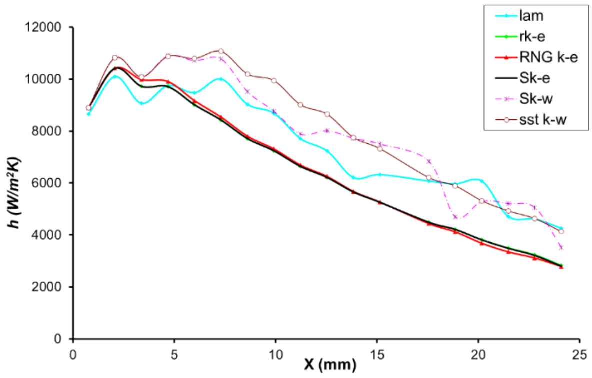


Figure 3: Profiles *h* coefficient "front"

(e = 1.2mm et ṁ = 0.1kg/min)

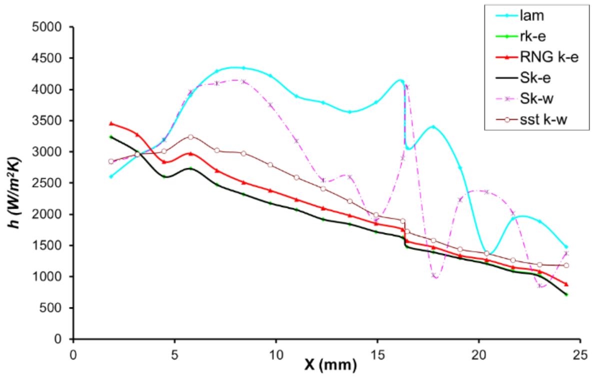


Figure 4: Profiles h coefficient "back"

(e = 1.2mm et ṁ = 0.1kg/min)

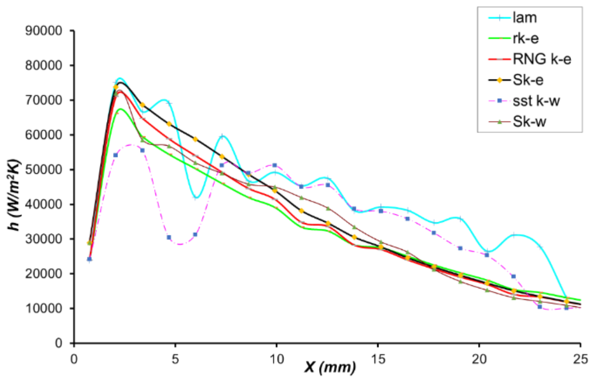


Figure 5:Profils h_{local} "front" (*e = 0.15 mm et ṁ = 0.9 kg/min*)

In this section we present results concerning the thermal field of the studied configurations. We will see the detailed impacts of the geometry dimension presented by the fluid passage (hydraulic diameter), the fluid acceleration intended by velocity used to evaluate *Re* and the wall reference temperature chosen to calculate the averaged thermal quantities (R_{th}, *h* and *Nu*).

Figure (6) shows thermal resistance profiles verses flow rate for two dimensions of fluid passage (0.*7mm* and 0.*35mm*). The results show, as expected, that the thermal resistance decreases with flow rate and then rises with fluid passage expansions, contrary to convective coefficient which increases with the flow rate and then drops with fluid passage enlargements, as it can be seen in figures (6 and 7).

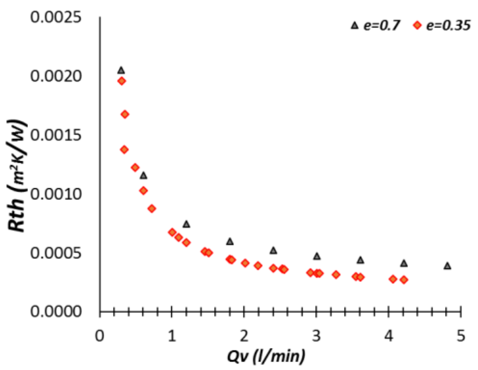


Figure 6: Thermal resistance vs *Qv (l/min)*

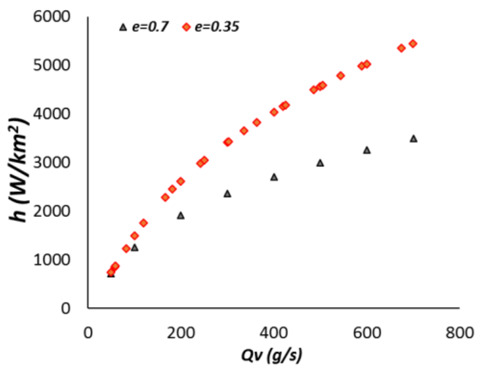


Figure 7: Convective coefficient vs *Qv (l/min)*

While for Nusselt number, increases with the increasing of flow rate and various fluid passage (*e=0.35, 0.7, 1.05, 1.4mm*), as it is shown in Figure (7), with Navier-Stocks and model. We can see that the realizable k-ε give better prediction of Nu with the increase of channel width (*Dh*) .

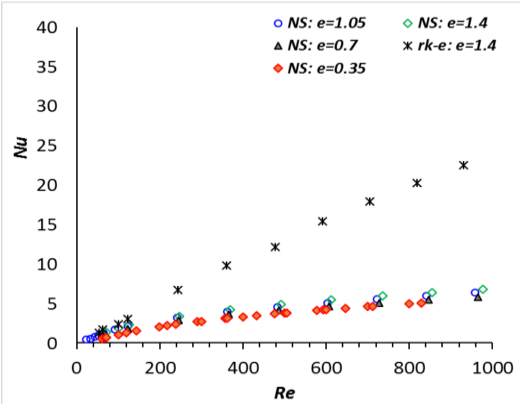


Figure 7: Nusselt comparison vs Re_{ave}/e

5. Conclusion

A comparison of different turbulence models was carried out, to select the most appropriate model for the configurations under study. For this purpose, we relied on various numerical simulation tests for configurations with different fluid passages. Various factors enabling us to judge the superiority of the k-ε family of models were then set out, justifying the choice and selection of the appropriate model with which to continue the study.

The k-ω models or any other RANS model showed a significant discrepancy. The anomaly is indeed due to the closeness of the walls when it comes to flow in mini/micro-channels, i.e. the influence of very narrow fluid passage (near-wall flow effects).

As the "Realizable" k-ε model was established to improve these results by introducing an improved near-wall treatment, its effects are very satisfactory for our case study.

Consequently, the mesh test has shown that continuing to over-refine the mesh is likely to give erroneous results, due to the amplification of truncation errors caused by over-refinement of the mesh (numerical solution explosion).

The best solution, therefore, is to apply the improved near-wall processing function, which doesn't require too much mesh refinement and saves a tremendous amount of computing time and memory space.

We conclude that the "Realizable" k-ε model combined with the "Enhanced Near wall treatment" method is the best for predicting flow in this type of geometry.

The thermal performance of this small-scale configuration is extremely close to the correct evaluation of the key conjugate heat transfer parameters. It is shown that the effects of the choice of reference velocity and temperature of the fluid and on the wall side are key factors. And the door is open to adequately calculate the convective coefficient *h*, the Nusselt number and the thermal resistance of the system, for a rigorous approximation of these parameters in such a reduced and compact geometry.

References

1. R.P., D.B. Tuckerman, "High-performance heat sinking for VLSI," IEEE ELECTRON DEVICE LETERS, vol. 2 (5), pp. 126-129, 1981.
2. J P Holman, Heat Transfer, SI Metric Editions, McGRAWHILL, BOOK COMPANY,1989.

3. K.A. Moores, Y.K. Joshi, "Effect of tip clearance on the thermal and hydrodynamic performance of a shrouded pin fin array" J. Heat Transfer, vol. 125, no. 6, p. 999–1006., 2003. E.M. Cherry, C.J. Elkins and J.K. Eaton. Geometric sensitivity of three-dimensional separated flows. *Int. J. of Heat and Fluid Flow*, 29: 803-811, 2008
4. Zukauskas, A. (1972). Heat transfer from tubes in crossflow. Advances in Heat Transfer, 8: 93-160. https://doi.org/10.1016/S0065-2717(08)70038-8
5. Moores, K.A., Joshi, Y.K. (2003). Effect of tip clearance on the thermal and hydrodynamic performance of a shrouded pin fin array. J. Heat Transfer, 125(16): 999- 1006. 99-1006. https://doi.org/10.1115/1.1621897.
6. Shkarah, A.J., Sulaiman, M.Y.B., Ayoba, M.R.B.H., Togun, H. (2013). A 3D numerical study of heat transfer in a single-phase micro-channel heat sink using graphene, aluminum and silicon as substrates international Communications in Heat and Mass Transfer, 48: 108-115. https://doi.org/10.1016/j.icheatmasstransfer.2013.08.006
7. Kandlikar,S.G. and Grande, W.J.,"Evaluation of single-phase flow in microchannels for high flux chip cooling e thermo-hydraulic performance enhancement and fabrication technology," Heat Transfer Eng., vol. 25, no. 8, pp. 5-16, 2004.
8. Kosar, A. and Peles,Y. "Thermal-hydraulic performance of MEMS-based pin fin heat sink," J. Heat Transfer, vol. 128, no. 2, p. 121–131.
9. Morini, G.L., "Single-phase convective heat transfer in microchannels: a review of experimental results," International Journal of Thermal Sciences, vol. 43, p. 631–651.
10. Rebay, M., Kakac, S., Cotta, R.M., (Eds.), (2016). Microscale and Nanoscale Heat Transfer: Analysis,Design, and Application (1st ed.). CRC Press.https://doi.org/10.1201/b192
11. Ahmadian-Elmi M. [2021], Mashayekhi A., Nourazara S.S., Vafaic K., " A comprehensive study on parametric optimization of the pin-fin heat sink to improve its thermal and hydraulic characteristics," International Journal of Heat and Mass Transfer Volume 180, December 2021, 121797, https://doi.org/10.1016/j.ijheatmasstransfer.2021.121797
12. Begag, A., SAIM R., ABBOUDI S., "Analyse numérique du comportement thermo-convectif d'un écoulement turbulent dans un mini canal en presence des ondulations en forme V," in Third International Conference on Energy, Materials, Applied Energetics and Pollution., Constantine, Algeria, ICEMAEP2016, October 30-31, 2016.
13. Al-Haddad, W., "Thermique des mini-canaux : comportement instationnaire et approche convolutive," Nancy, France : Lab. lemta, university of Lorraine, PhD Thesis, HAL Id: tel-01468830
14. Bessanane, N., Si-Ameur, Hamza A., Daguerre J.P., M., Rebay, M., "Numerical and Experimental Study of the Fluid Flow in Diamond Shapped Heat Sink," The international symposium on Convective Heat and Mass Transfer, Kusadasi, Turkey, CONV-2014, june 8-12, 2014.
15. Bessanane, N., "Simulation numérique de la convection turbulente : géométries régulières et complexes," Reims, France : Lab. ITHMM, university of Reims, PhD Thesis, 2018, https://www.theses.fr/235615595
16. Bessanane, N., Si-Ameur, M., Rebay, M., "Numerical Study of the Temperature Effects on Heat Transfer Coefficient in Mini-ChannelPin-Fin Heat Sink,"International Journal of Heat and Technology, Vol. 40, No. 1, February, 2022, pp. 247-257, https://doi.org/10.18280/ijht.400129
17. Debray F., Franc J.P., Maître T, Reynaud S., " Mesure des coefficients de transfert thermique par convection forcée en mini-canaux," Mechanics & Industry, EDP Sciences, 2001, 2, pp.443-454. HAL Id: hal-00212312
18. YANG Li, J., and Huazhi LI, W. "2-Dimensional CFD Simulation and Correlation Development for Optimization of Fin Heatsinks in Electronic Cooling," J. of Thermal Science, vol.10, (4), p. 363-371.
19. Jiang, P.X., and Xu,R.N, "Heat transfer and pressure drop characteristics of mini-fin structures,,"
20. Liu, M.H., Liu, D., Xu, S., Chen, Y.L. (2011). Experimental study on liquid flow and heat transfer in micro square pin fin heat sink. International Journal of Heat and Mass Transfer, 54(25-26): 5602-5611. https://doi.org/10.1016/j.ijheatmasstransfer.2011.07.013