COOLING OF A SURFACE WITH CONSTANT HEAT FLOW USING A SYNTHETIC JET ACTUATOR

Embarek DOUROUM[1], Samir LAOUEDJ[2], Abdelylah BENAZZA[3], Amar KOUADRI[4]

The present paper examines the sensitivity of synthetic jets used for thermal enhancement in a micro-channel. The fundamental concentration in this paper is the examination of the influence of the cavity diaphragm position on the behavior of the flow produced by the jet and on the heat transfer enhancement. The proposed hybrid system contains an actuator of a synthetic jet mounted in a micro-channel for the cooling of heated surface placed also in the micro-channel. The flow in the microchannel is modeled by a 2D simulation to solve the URANS equations with finite volume discretization, the turbulence model k-ω (SST) was used to modeling the jet flow. This study makes a comparison between the different configurations studied about the vortex structures generated by the jet, velocity contours and heat transfer characteristics, which permits us to know that there an effect of these studied modifications on the heat transfer improvement, where there is an improvement of the thermal cooling caused by the modification of the nature of the flow in the micro-channel and the interaction between the transversal flow and the vortex structures generated by the jet actuator, in particular for the proposed case.

Keywords: Heat transfer improvement, Cooling, Transversal flow, Microchannel, Synthetic jet cavity.

1. Introduction

Synthetic jets are fluidic devices which have imposed themselves in several fields among them, the heat transfer applications. They can be used for laptop microprocessor cooling, light emitting diode (LED) cooling, and cooling of very high power plate on board type LEDs [1]. The interaction of successive vortices coming out of the cavity through an orifice able to create the synthetic jet. Synthetic jets are produced by a periodic motion of a piston or an oscillating

[1] Researcher., Laboratory of materials and reactive systems, University of Sidi bel Abbes, Algeria, e-mail: embarek.douroum@univ-sba.dz.
[2] Lecturer., Laboratory of materials and reactive systems, University of Sidi bel Abbes, Algeria, e-mail: samirladz@yahoo.fr
[3] Prof., Laboratory of materials and reactive systems, University of Sidi bel Abbes, Algeria, e-mail: abdel.benazza@gmail.com
[4] Lecturer., Laboratory of developement in mechanics and materials, University of Djelfa, Algeria, e-mail: kouadriamar@yahoo.fr

diaphragm within a cavity containing one or more orifices in its other walls. Synthetic jets have a preference over continuous jets, they don't need an external supply fluid, where the amount of fluid aspirated from the flow is returned to the flow again [2].

Many concerns have been treated by researchers interested in the thermal design of jet impact systems such as: the improvement of the convective heat transfer, trying to cover the largest area of the impingement surface, the augmentation of the rate of the heat transfer and the reduction of the mass flow rate of the used coolant [3, 4]. The increase of the convective heat exchanges using passive and active methods by impacting jets has been vastly studied, like exciters of turbulence (protuberances) [5], different nozzle shapes [6, 7], increasing of surfaces exposed, vortex creators [8], acoustic flow excitation [9] and synthetic jet [10].

The formation of the synthetic jet and its evolution have been studied experimentally by Smith et al. [2], they described the behavior of a jet through a rectangular opening. They have made a comparison between a synthetic jet and an ordinary 2D jet, their results show that the synthetic jet benefits well from the environment flow, but it loses his momentum quickly vis-a-vis the continuous jet.

A lot of studies have been realized to know the importance of using synthetic jets to cooling the heated surfaces. Campbell et al. [10] have shown that the synthetic air micro-jets were efficient in the laptop processors cooling. They found that on average, a 22% to 26% reduction in the temperature of laptop processors was obtainable with synthetic jets vis-a-vis the cooling by natural convection methods. While Mahalingam et al. [11, 12] developed an application using the actuators of jet and the fan. They found that the jet can dissipate more power than a fan mean. Smith et al. [13], as well as Pavlova et al. [14] realized an experimental study on the impact of synthetic jets for a heated wall cooling, they made a comparison the synthetic jet performance with the continuous jet performance. They concluded that the synthetic jets ensure more cooling arrives at three times more efficient than the continuous jets cooling.

The thermal performance of the synthetic jet has not been sufficiently studied numerically, and the studies published in this context remain limited. Timchenko et al. [15] studied numerically the thermal improvement in a 200 mm high micro-channel using an oscillating membrane. They studied, by comparison, the efficiency of this process of cooling with and without jet. For that, they used a factor of heat transfer effectiveness, where this factor increased for the case of the synthetic jet by 64%. Though the importance of the turbulence in synthetic jet flows, they didn't use a model of turbulence in their simulations. In another numerical investigation, Erbas et al. [16] performed a numerical investigation of the use of an actuator of a synthetic jet in a 2D channel to evaluate its heat transfer efficiency on a heated wall. In this study, they designed different synthetic jet actuators to study their efficiency in thermal enhancement. The number of actuators, the location, and the phase shift of the membranes are the parameters tested. Their results showed that the rate of heat transfer increases with the number of jets, the proper location, the use of the nozzle throat shape and the phase shift of 180° between the two jets. But also, they didn't study in their investigations the effect of the transversal flow on the thermal performance. For facilitate the computation domain studied, authors (Kral et al. [17]) have minimized it by modeling only the zone outside the actuator cavity, while, Rizzetta et al. [18] have a numerical study covers all the flow fields. Mallinson et al. [19, 20] also have studied numerically and experimentally the total flow field (all the actuator). Laouedj et al. [21] have analyzed numerically the effect of an actuator orifice obstruction on the thermal characteristics and on the interaction between the jet and the cross-flow, their obtained results show that an obvious thermal enhancement can be found usin the obstruction especially for the cases with 45 and 60 % obstruction and with an oscillating amplitude below 50 um. An other numerical comparative study has been carried out by Benayad et al. [22], they used the same configuration as used above but with two cavities and with an undulation heated surface, they found that the Nusselt number has been increased by 51 % compared with the basic case (with a single cavity).

The study of flows inside the cavity remains limited in published works, so the details in the cavity must be better documented. Even for the design of the orifice, it has not been sufficiently studied, particularly its shape and with obstruction. In this study, we tried to examine the sensitivity of the design of the actuator cavity of the synthetic jet using the computational fluid dynamics (CFD). The simulations focused exactly on the position of the oscillating diaphragm in the cavity walls. This study compares the eddy structures emitted during the blowing phase, the velocity profiles and other dynamic jet characteristics for different cases, which permits us to understand the influence of the modifications on the thermal improvement. We hope that this study is useful for the design of these devices and their development to achieve appropriate models.

2. Materials and methods

2.1. Cavity designs

The two different configurations of the actuator designs examined in this study are shown in Fig. 1 and Fig. 2.

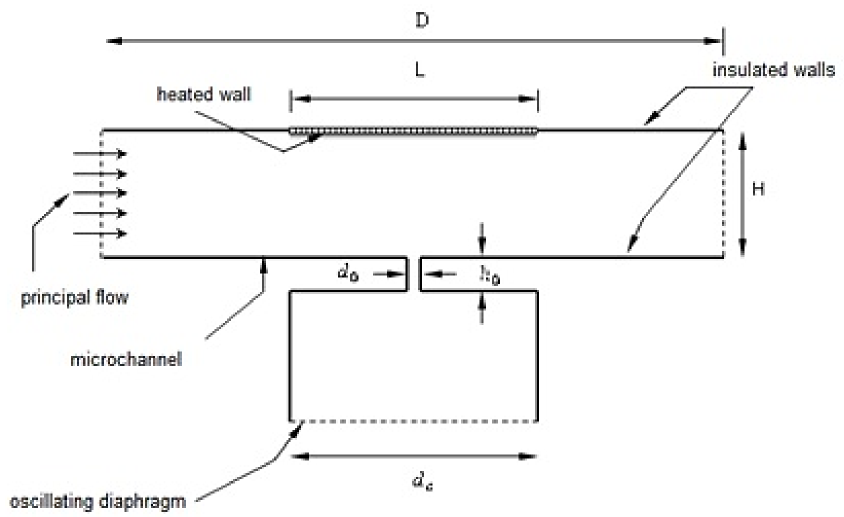

Fig. 1. Configuration of the first case

The first configuration represents a cavity with aspect ratio L/d_0=15, its diaphragm is the bottom wall of the cavity. This configuration has been studied previously by Chandratilleke et al. [23].

The second configuration represents a cavity which has the same shape as the first configuration except its diaphragm is both the cavity side-walls.

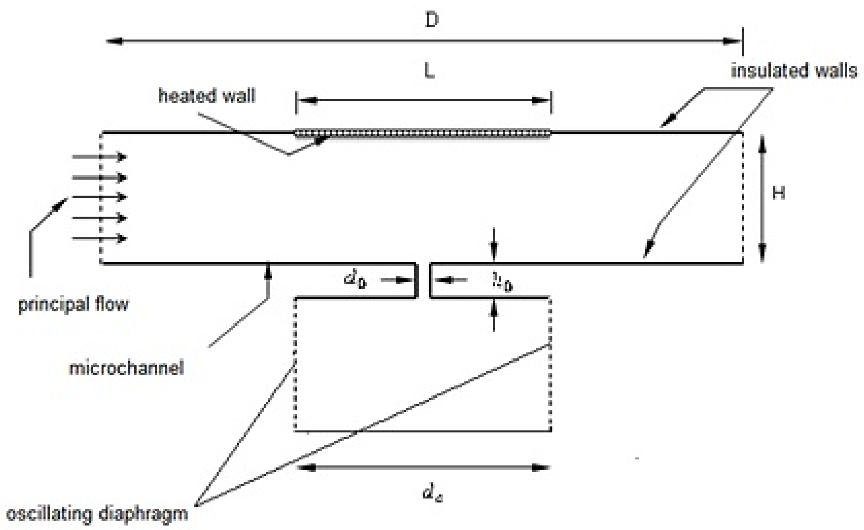

Fig. 2. Configuration of the second case

2.2. Numerical model

This study contained a numerical simulation to examine the thermal and the hydrodynamic behaviour of a turbulent fluid jet with low Reynolds numbers interacting with a transversal flow in a micro-channel in order to cooling a heated wall. The equations governing the flow are the continuity equation, the momentum equations and the equation of energy:
The continuity equation:

$$\frac{\partial u_i}{\partial x_i} = 0 \tag{1}$$

The momentum equation is expressed by:

$$\rho\left(\frac{\partial u_i}{\partial t} + u_j \frac{\partial u_i}{\partial x_j}\right) = -\frac{\partial p}{\partial x_j} + \mu \Delta u_i \tag{2}$$

At each time step, the spatially averaged instantaneous Nusselt number can be calculated by the following expression:

$$Nu(t) = \frac{q}{\overline{T_w}(t) - T_b(t)} \frac{L}{k} \tag{3}$$

Where $\overline{T_w}$ presents the spatially averaged temperature, and T_b is the bulk flow temperature, calculated using the approach of the mass-weighted average, while L is the heater width, k is the fluid thermal conductivity and q is the heat flux.

2.3. Mesh generation

A structured mesh (quadratic cells) was used for the entire domain, with intensity near the walls in order to capture the complex details concerning the formation and the separation of the jet, the density of the mesh around the orifice was refined to fourteen cells in the axial direction and twenty in the transverse direction. The model has 48072 elements.

2.4. Boundary conditions

Table 1 and Fig. 3 show that the used fluid (air) in our study was supposed as an incompressible fluid and an isothermal substance. We took the thermodynamic properties of the fluid at 300 K at atmospheric pressure. A constant heat flux of 15 KW/m^2 was imposed in the heated top surface, it is the wall where the impingement of the jet occurs.

Table 1

Thermodynamic properties of the fluid

Property	Density $\boldsymbol{\rho}$	Specific heat C_p	Viscosity $\boldsymbol{\mu}$	Thermal conductivity k
Hypothesis	Ideal gas	Constant	Constant	Constant
Value	1.1614 e-03 g/cm^3	1.007 e-03 KJ/g k	1.7894 e-05 Pa.s	0.263 e-03 W/cm k

The diaphragm motion was described by a sinusoidal function, which has been programmed by the *C* language. As an algorithm of the solution, we used the segregated resolution method with an implicit solver formulation. And concerning the momentum, pressure, density, the kinetic energy of the turbulence and the specific dissipation rate we used the second-order discretization schemes, while the pressure is coupled with the velocity using the (PISO) algorithm. In our investigation, the flow of the jet happens periodically by the diaphragm oscillating inside a confined region. The generation of the jet can cause the appearance of the turbulence in some regions in the field, even the flow always remains in laminar conditions, because the Reynolds number values remain low according to the boundary conditions. To modeling the jet interaction with the transversal flow, the k–ω (SST) turbulence model was used. We selected this model because it has the advantage of giving a good prediction in the area near the wall, and also far from that. For these conditions, the URANS and the energy equations were solved in the computational domain, for a group of operating parameters. The simulation was realized by dividing each cycle into 36 time steps, and in each time step, 20 sub-iterations were performed. The simulation was carried out for 20 cycles to obtain a stable convergence of the computation.

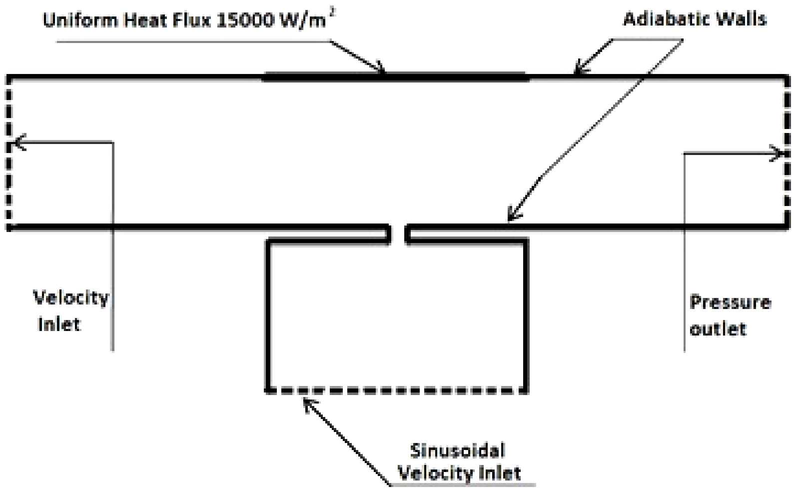

Fig. 3. Boundary conditions

2.5. Simulation conditions and methodology of solving

Oscillating diaphragm movement was modeled by a piston motion by a dynamic mesh technique. In order to describe the periodical motion of the oscillating diaphragm, a user-defined function (UDF) has been written in the C language program and incorporated into the solver of FLUENT. The initial position of the diaphragm (when: t = 0) is considered at the bottom of the cavity. The function of the diaphragm displacement is defined as:

$$y = A\, sin(\omega t - \varphi) \tag{4}$$

Where y is the diaphragm position (when, $y = 0$, the cavity has a maximum size and if $y = A$, the cavity has a minimum size), A represents the diaphragm amplitude, while ($\omega = 2\pi f$) represents the angular velocity, with f symbolizes the oscillating diaphragm frequency. And finally φ represents the angular phase shift that permits the diaphragm to begin its motion from the cavity base.

Before starting a numerical simulation with the Fluent software, we must provide an initial condition from which the software starts the calculation. The appropriate choice of initial conditions can achieve a stable solution and faster convergence. The initialization of the calculation can be performed from the entry, exit or from the entire domain. In our case, we chose to initialize the calculation from all zones because a lot of values of physical quantities are known. At every cycle time step, the calculate iterations continued till the residual parameters of mass, momentum and turbulence (kinetic energy k and specific dissipation ω) were decrease it below 10^{-3}, while 10^{-6} for the energy residue. Unsteady simulations converged periodic solution from the 18th period to analyze in some stable operation of the actuator. In order to ensure the precision of equations solving, we used the solver double precision 2D.

2.6. Characteristics of heat transfer

Numerical investigation of the cooling phenomenon of a wall placed in a micro-channel by a synthetic jet actuator was conducted using a statistical method unsteady RANS modeling of turbulence. In heat transfer operations, many dimensionless numbers are used, among them, we have the Nusselt number. It reflects the heat exchange quality:

$$Nu = \frac{hL_{char}}{k} \tag{5}$$

Where: L_{char}: presents the characteristic length, which depends on the studied geometry. For the pipe flow case, consider it the diameter of the pipe, or if the hydraulic diameter of the conduit does not have a circular section. For the flat plate, it will take the plate length or from the abscissa of the leading edge plate. Like any dimensionless number, the value of the Nusselt number depends heavily on reference variables that are chosen, and the physical meaning is meant to give it (eg local or global). It is particularly important to know, when using a correlation, if the coefficient of convection h was defined with respect to a fixed reference temperature, or a temperature of the local mixture

3. Results and discussion

3.1 Model validation

The validation of the model in this study was carried out by comparing our results obtained with those of Chandratilleke et al. [23] keeping the same physical and geometric conditions, the evolution of the local Nusselt number along the heated surface for the value of the inlet velocity (Vi= 1 m/s) is presented in figure 4. This figure shows that our results are in good agreement with those de Chandratilleke et al.

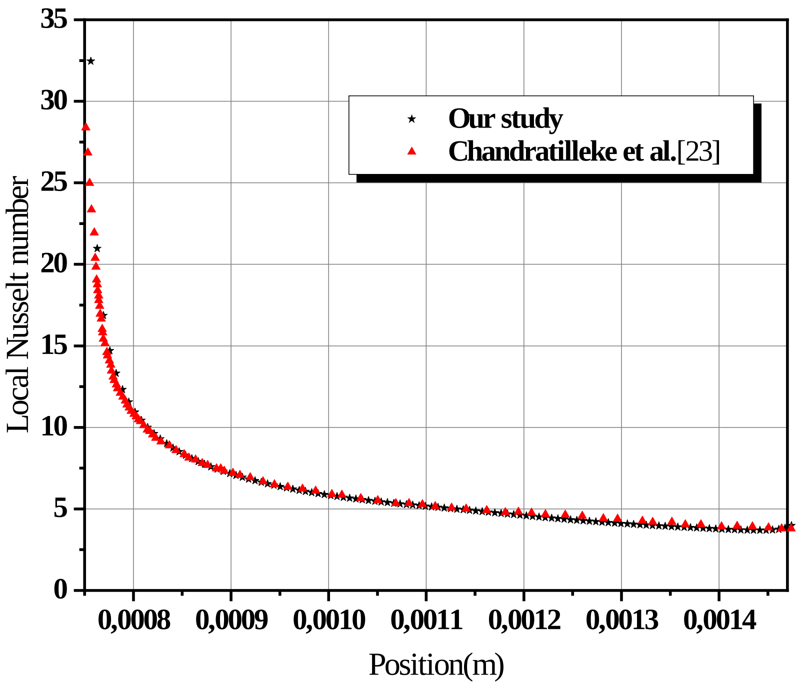

Fig. 4. Evolution of the local Nusselt number along the heated surface for Vi = 1m/s

3.2 Velocity and characteristics of the fluid flow

Figure 5 shows that there is a thermal improvement for the case 2 vis-a-vis the first case, this was reflected by the evolution of the heat exchange coefficient, this coefficient is observed to be higher and more intense in the middle of the heated wall due to the vortex binaries in interaction with the low velocity of the cross-flow.

Figures 6 and 7 illustrate the instantaneous contours of velocity in the computational domain, under the following conditions: the amplitude of the diaphragm ($50\mu m$), the inlet velocity in the micro-channel ($0.5\ m/s$) and the diaphragm frequency ($10Khz$). They illustrate the nature of the occurring encounter in the micro-channel between the transversal flow and the pulsating jet in the micro-channel over one operation cycle. During the upward movement of the diaphragm, a fluidic jet is blown via the cavity orifice with an elevated velocity into the micro-channel flow.

By the oscillating movement of the diaphragm with given amplitude, a sufficient fluid momentum is strongly injected in the transversal flow in the micro-channel, it allows the jet to break through the channel flow to amount to the hot surface especially until the moment $t = T/2$ (the end of the blowing phase). These figures show clearly the forming of the vortex structures of the synthetic jet throughout the period first half. We observe that the style of the flow is asymmetric, which is provoked by the flow drag transported by the micro-channel flow.

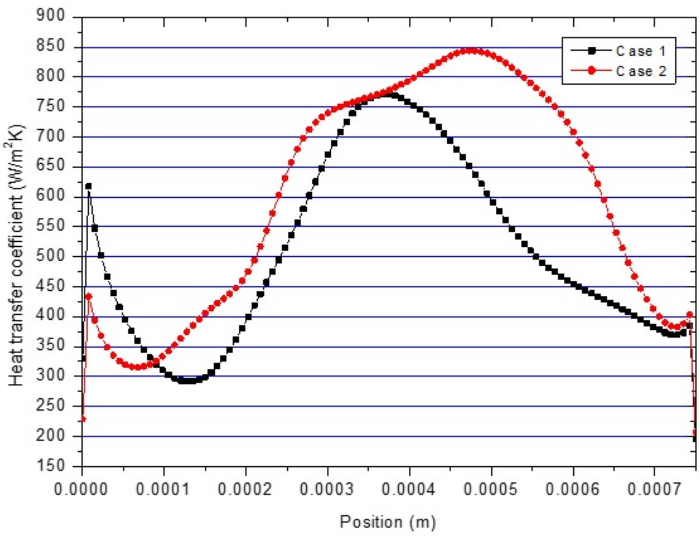

Fig. 5. Heat transfer coefficient at the end of the blowing phase for two cases With: A= 75.10^{-6} m, f =10000 Hz and Vi=0.5 m/s

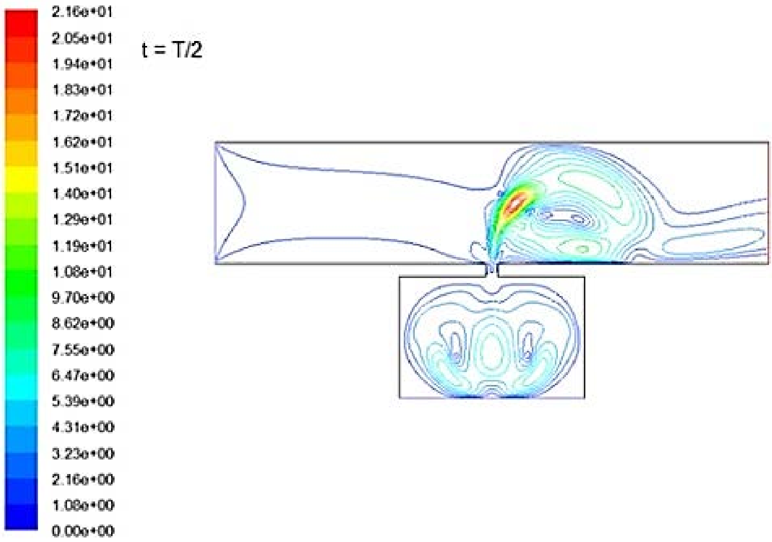

Fig. 6. Velocity contours for the first case at the end of the blowing phase (t=½T)

With: A= 50.10^{-6} m, f =10000 Hz and Vi=1 m/s

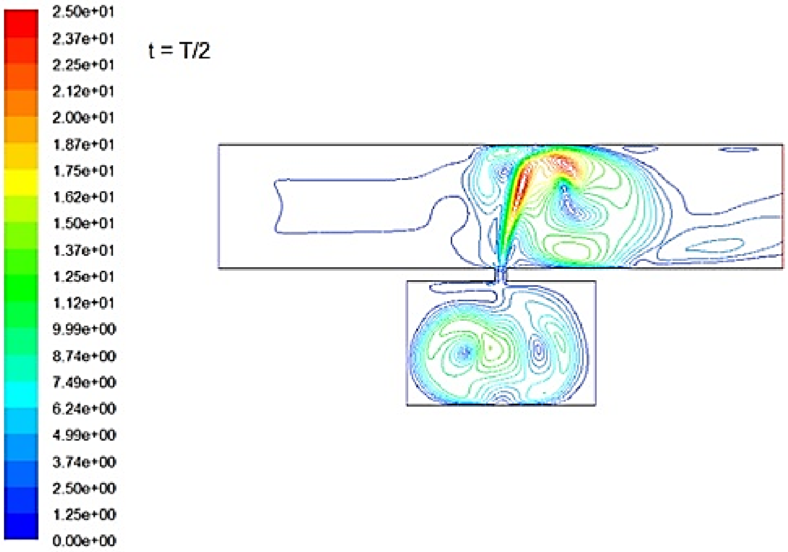

Fig. 7. Velocity contours for the second case at the end of the blowing phase (t=½T)

With: A= 50.10^{-6} m, f =10000 Hz and Vi=1 m/s

3.3 Vorticity

In this part, we describe the behavior of the vortex observed in our studied cases. In figures 8 and 9, the construction of the synthetic jet vortices is distinctly visional throughout the blowing phase.

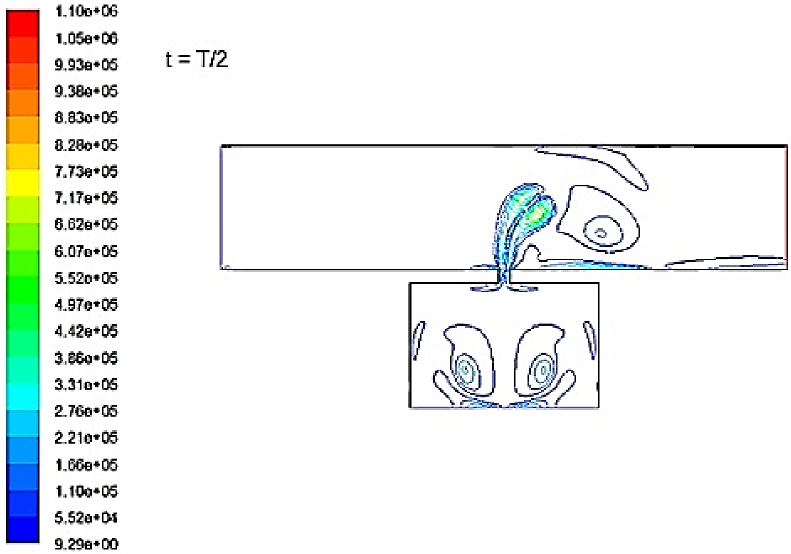

Fig. 8. Vorticity contours for the first case at the end of the blowing phase (t=½T)

With: A= 50.10^{-6} m, f =10000 Hz and Vi=1 m/s

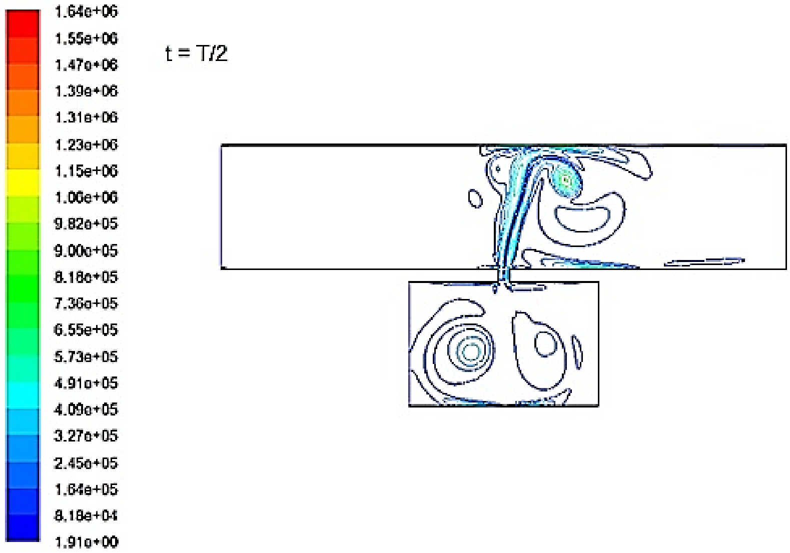

Fig. 9. Vorticity contours for the second case at the end of the blowing phase (t=½T)

With: A= 50.10^{-6} m, f =10000 Hz and Vi=1 m/s

From $t = T/2$, the diaphragm completes its cycle by relegating and starting the second phase of sequance. During this second half of the cycle, the fluid is pulled towards the cavity provoked by the diaphragm descending motion. At this moment, the vortex structures of the synthetic jet formed in the first phase are advected downstream by the micro-channel cross flow. The momentum produced by the synthetic jet reaches periodically the heated surface breaks the hydrodynamic and thermal boundary layers of them

3.4 Temperature distributions

The temperature contours in Figure 10 show clearly the synthetic jet mechanisms. The zones with the high thermal gradients are represented by the regions near the heated surface that have high temperatures. We can understand from these figures the interaction effect of the transversal flow in the microchannel and the vortices of the synthetic jet produced by the jet actuator during one cycle. At the end of the blowing phase, the diaphragm starts to descend after arriving its top in order to terminate the cycle.

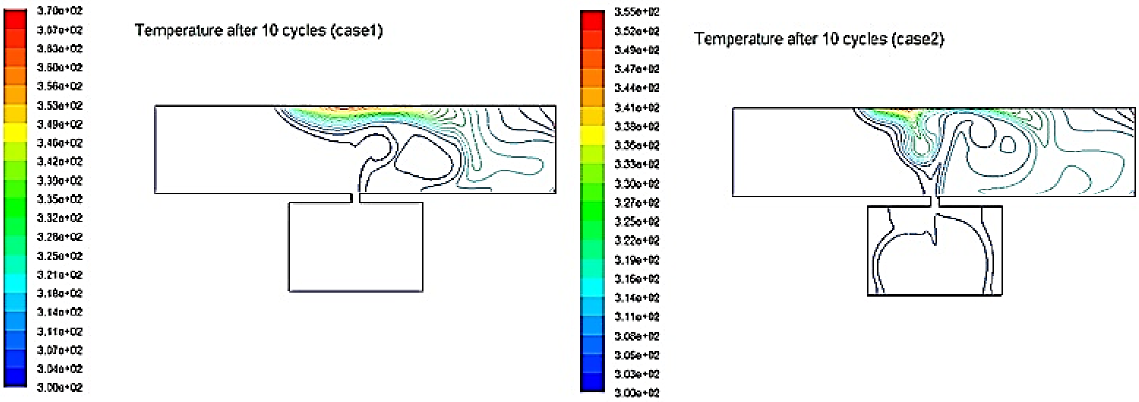

Fig. 10. Temperature contours for the case 1 and case 2, after 10 cycles with: A= 50.10^{-6} m, f =10000 Hz and Vi=1 m/s

During the aspiration phase, the fluid jet will enter into the cavity. The zone of high thermal gradient near the heated surface seems smaller for the second case vis-a-vis the first case, this region is influenced by the vortices generated by the oscillatory movement of the diaphragm in both cases, chiefly in the second case. Therefore, the oscillatory flow mechanisms can improve the heat exchange in the assemblies equipped by an actuator of a synthetic jet.

4. Conclusions

We realized these numerical simulations in order to study the influence of the design of the jet actuator cavity on the synthetic jets behaviour. The designing changes discussed here is the modification of the oscillating diaphragm location. A synthetic jet crossed with the transversal flow was studied. As a general conclusion, the modifications in the design of the jet cavity have a relatively limited influence on the flow at the exit of the jet. During the cycle phases, we can observe considerable disparities throughout the generation of the jet. The asymmetric flow in the jet cavity creates an asymmetric vortex binary in the crossflow, this binary undergoes a horizontal deviation towards the downstream influenced by the transversal flow. These vortices maybe have an important role in the electronic components cooling by the impact of fluidic jets, where it's necessary to target the heated surfaces by a fluidic jet.

Obtained results are important for the designing, but on the other hand, the details of the design of the jet cavity do not affect directly and individually the jet performances. Therefore, these parameters can be utilized for design modifications to achieve satisfying results and without negatively affecting the jet performances. This study of the approximate modeling of the jet actuator may be sufficient if the main interest of this modeling is the prediction of the external flow of the jet.

In our study we have two caveats, the first is that our treated parameters in this investigation are limited, so we can't derive an inclusive conclusion by this study. Secondly, in this study, the effects of the compressibility are not taken in account, while a compressible flow in a cavity is more sensitive to the design of them. So we need to investigate more this influence numerically and experimentally. For case 2, the synthetic jet interacts with the transversal flow in the micro-channel, that interaction gives an evident thermal improvement at the heated surface.

R E F E R E N C E S

[1]. *A. Arshad, M. Jabbal,Y.Yan,* “Synthetic jet actuators for heat transfer enhancement – A critical review”, International Journal of Heat and Mass Transfer, **Vol. 146**, 2020, 118815

[2]. *B. L. Smith, A. Glezer*, “The formation and evolution of synthetic jets”, Physics of Fluids, **vol. 10**, 1998, pp. 2281-2297.

[3]. *H. Martin,* “Heat and mass transfer between impinging gas jets and solid surfaces”, Advanced Heat Transfer, **vol. 13**, 1977, pp. 1-60.

[4]. *K. Jambunathan, E. Lai, M. A. Moss, B. L. Button*, “A review of heat transfer data for single circular jet impingement”, International Journal of Heat and Fluid Flow, **vol. 13**, 1992, pp. 106-115.

[5]. *P. Hrycak,* “Heat transfer from impinging jets to a flat plate with conical and ring protuberances”, International Journal of Heat and Mass Transfer, **vol. 27**, 1984, pp. 2145-2154.

[6]. *A. M. Huber, R. Viskanta,* "Impingement heat transfer with a single Rosette nozzle", Experimental Thermal Fluid Science**, vol. 9**, 1994, pp. 320-329.
[7]. *D. W. Colucci, R. Viskanta*, "Effect of nozzle geometry on local convective heat transfer to a confined impinging air jet", Experimental Thermal Fluid Science, **vol. 13**, 1996, pp. 71-80.
[8]. *N. Gao, H. Sun, D. Ewing,* "Heat transfer to impinging round jets with triangular tabs", International Journal of Heat and Mass Transfer, **vol. 46**, 2003, pp. 2557-2569.
[9]. *S.D. Hwang, C.H. Lee, H.H. Cho,* "Heat transfer and flow structures in axisymmetric impinging jet controlled by vortex pairing", International Journal of Heat and Fluid Flow, **vol. 22**, 2001, pp. 293-300.
[10]. *J.S. Campbell, W.Z. Black, A. Glezer, J.G. Hartley,* "Thermal Management of a Laptop Computer with Synthetic Air Microjets", in IEEE InterSociety Conference on Thermal Phenomena, **vol. 6**, 1998, pp. 43-50.
[11]. *R. Mahalingam, A. Glezer,* "Air Cooled Heat Sinks Integrated with Synthetic Jets", in IEEE In InterSociety Conference on Thermal Phenomena, **vol. 8**, 2002, pp. 285-291.
[12]. *R. Mahalingam, N. Rumigny, A. Glezer,* "Thermal management using synthetic jet ejectors", IEEE Transactions on Components and Packaging Technologies, **vol. 27**, 2004, pp. 439-444.
[13]. *B. L. Smith, G. W. Swift,* "A comparison between synthetic jets and continuous jets", Experiments in Fluids, **vol. 34**, 2003, pp. 467-472.
[14]. *A. Pavlova, M. Amitay,* "Electronic cooling using synthetic jet impingement", Journal of Heat Transfer, **vol. 128**, 2006, pp. 897-907.
[15]. *V. Timchenko, J. Reizes, E. Leonardi,* "A numerical study of enhanced microchannel cooling using a synthetic jet actuator", In Proc 15^{th} Australasian Fluid Mechanics Conference, Sydney, Australia, 2004.
[16]. *N. Erbas, O. Baysal,* "Micron-level actuators for thermal management of microelectronic devices", Heat Transfer Engineering, **vol. 30**, 2009, pp. 138-147.
[17]. *L. D. Kral, F.D. John, A.B. Cain, W. C. Andrew,* "Numerical simulation of synthetic jet actuators", American Institute of Aeronautics and Astronautics, **vol. 4**, 1997, pp. 1-14.
[18]. *D. P. Rizzetta, M. R. Visbal, M. J. Stanek,* "Numerical investigation of synthetic jet flow fields", American Institute of Aeronautics and Astronautics, **vol. 37**, 1999, pp. 919-927.
[19]. *S. G. Mallinson, J. A. Reizes, G. Hong,* "An experimental and numerical study of synthetic jet flow", The Aeronautical Journal, **vol. 105**, 2001, pp. 41-49.
[20]. *S. G. Mallinson, C. Y. Kwok, J. A. Reizes,* "Numerical simulation of micro-fabricated zero mass-flux jet actuators", Sensors and Actuators A, **vol. 105**, 2003, pp. 229-236.
[21]. *S. Laouedj, J. P. Solano, A. Benazza,* "Synthetic jet cross-flow interaction with orifice obstruction", International Journal of Numerical Methods for Heat & Fluid Flow, **Vol. 25**, 2015, pp. 749-761.
[22]. *Z. Benayad, S. Laouedj, A. Filali,* "Numerical investigation on the cooling of electronics components with synthetic multi-jets and non-sinusoidal bi-periodic forcing functions", Energy Reports, **vol. 6**, 2020, pp. 1-9.
[23]. *T. T. Chandratilleke, D. Jagannatha, R. Narayanaswamy,* "Heat transfer enhancement in microchannels with cross-flow synthetic jets", International Journal of Thermal Sciences, **vol. 49**, 2010, pp. 504-513.