Improving the Cooling Process for Electronics using Synthetic Jets

Damian Gaona
Vaughn College of Aeronautics and Technology, United States, damian.gaona@vaughn.edu

Mentors: Amir Elzawawy, PhD
Vaughn College of Aeronautics and Technology, United States, amir.elzawawy@vaughn.edu

Hossein Rahemi, PhD
Vaughn College of Aeronautics and Technology, United States, hossein.rahemi@vaughn.edu

Abstract*– *The progressive increase of heat dissipation from modern electronics requires powerful cooling systems to improve the performance and reduce the thermal fatigue of electrical components. Currently, the attention on synthetic jet systems has increased due to the improvement of the motion flow performance. Often, engineers face different physical phenomena that cannot be described mathematically or by applying differential equations where it is long and complex to solve the problem. Therefore, an experimental program must be applied to solve the problem of the conductivity over a Newtonian fluid over a thermal boundary layer within an integrated synthetic jet system. The experimental program applied to solve the heat conductivity is COMSOL, using the computational fluid dynamics (CFD) module to define the Nusselt number. The Nusselt number represents the enhancement of heat transfer through a fluid layer. The larger the Nusselt number is, the more effective the convection of the heat transfer. In the simulation, different factors will be taken into consideration: the uniform flow velocity, location of the synthetic jet system, and the frequency of the piezo diaphragm. The study will present the heat transfer association with the natural convection driven by the temperature gradients in a Newtonian fluid.

Keywords: Synthetic jet, heat convection, heat conduction, thermal boundary layer, Nusselt number.

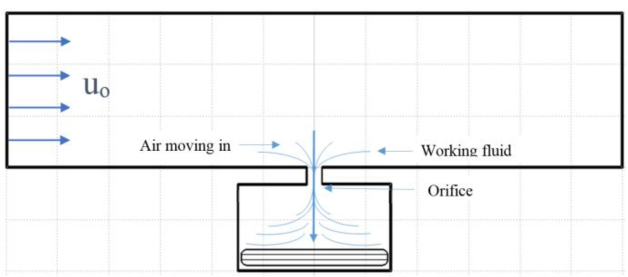
I. INTRODUCTION

The modern lifestyle of many people has become increasingly dependent upon microelectronic devices that support everyday needs, such as cell phones, laptops, navigation systems, tablets, and so on. The ever-expanding growth in technology demands the development of faster microprocessors and RAM. Because of this, the need for compact devices with better heat dissipation reduction is a current topic of interest among researchers. When operating these devices, significant levels of heat are created. This heat must be dissipated to ambient conditions in order to avoid accumulating heat within a closed system. The result of the increased temperatures is mainly thermal fatigue, whereby the internal heat builds up within electronic packages. This action raises the microchip temperatures above what is tolerable, and the excessive heat causes unrepairable damage or a complete thermal breakdown due to overheating with a reduction of the device's working life. The concept of the Nusselt number presents that the heat transfer through the solid plate is by conduction, and the heat transfer through the fluid flow is by convection. Therefore, the Nusselt number is the ratio of the convective over the conductive heat transfer.

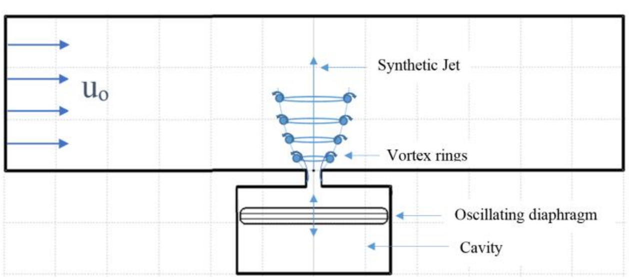
Of the causes of semiconductor device failures, "die-attach failure" is one of the six most common. In a 1999 article for EDN Network [1], V. Lakshminarayanan explains the process of die-attach failure: where the improper contact between the die and substrate influence over the thermal conductivity deceasing it in between the two. The results present that the die can overheat leading to an increment of stress and cracks, and in the end the device ends up failing. Efficient cooling systems are known to improve the capacity, performance, and the lifetime of devices; this in turn will make the user happy. Many cooling applications today exist to dissipate the heat efficiently and safely with minimal investment and maintenance. The most common cooling systems applied to the microelectronic devices are fans and liquids. The fans have a disadvantage. In order to increase the dissipation levels of a device, a higher fan speed is required, which increases noise and reliability issues. Currently, the common market desires lighter and thinner devices with better RAM and hardware. The problem is that these small but highly advanced pieces of hardware will produce a lot more heat. Since these devices are smaller and as such have a smaller surface area, there is a risk that the advanced RAM (which would generate a lot more heat than the RAM devices of older device models) will overheat much faster and substantially increase the likelihood of thermal fatigue failure. Many people today desire a cooling system that is able to perform well even in small areas. Different researches have been recently conducted regarding the improvement of heat transfer with synthetic jet systems. Many have obtained favourable results for the improvement of heat dissipation on devices.

II. SYNTHETIC JET ACTUATORS

A jet can be synthesized [2] [3] by forcing fluid in and out of an orifice or channel at pre-defined frequency by sending electric pulses to piezo material installed over a diaphragm. Synthetic jets introduce no net mass flow into a system; instead, they deliver flow with net positive momentum as they generate a train of vortex pairs that self-sustain an outward periodic flow. Figures 1a and 1b project a visualization of the system. The main difference of a synthetic jet with other jets is the fact that is a zero-mass flux system in nature and produce fluid flow with finite momentum with no mass addition to the system and without need of complex plumbing. When activated, the ability to direct airflow along heated surfaces in the confined environment. The small-scale mixing presents that the synthetic jet system is ideally suited for cooling applications for devices that requires effective heat dissipation while the device is in use. Different concepts of heat transfer and hydrodynamics are applied for the understanding of this system.



(a)

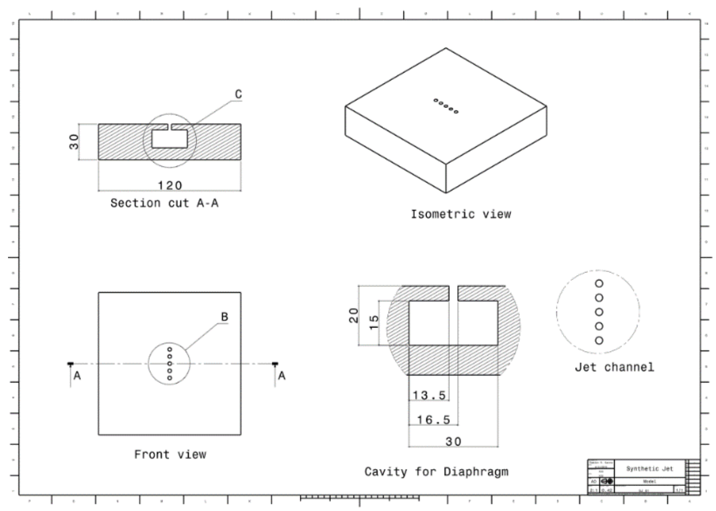


(b)

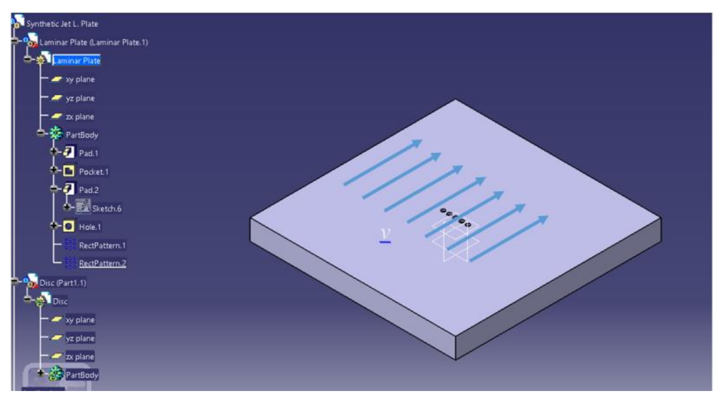
Figure 1: Structure of Synthetic Jet, a) Expansion stroke b) Compression stroke

Figure 1 presents the structure that produces a synthetic jet of the flow. The cycle of the piezo diaphragm is moving at a frequency with a sine equation for the oscillation of the diaphragm. The oscillated diaphragm is presented with the arrow that in the compression stroke will produce vortices that will be part of the synthetic jet ejected to the plate to improve the dissipation of the heat flux from the surface. Ideally, the goal is to evaluate the heat transfer convective coefficient of the plate with more than one channel to understand the influence of the number of channels. The location of the channel and the velocity of the forced flow over the plate.

Figures 2a and 2b present the CATIA V5 3D model of the laminar plate that will be simulated in a 2D field in COMSOL.



(a)



(b)

Figure 2: CATIA 3D ideal model for simulation, a) CATIA 3D model drafting (units [cm]), b) CATIA 3D model with direction of flow

The 3D model on CATIA in figure 2 subjected to a flow velocity (v); where it can be imported to COMSOL software for simulation.

III. GOVERNING EQUATIONS

From the heat transfer book applied in the course, it was presented that there are three basic mechanisms of heat transfer, conduction, convection and radiation [4]. In many ways, conduction and convection are similar since both mechanisms require the presence of the material medium. However, the difference is that the convection mechanism requires the presence of fluid motion, similar to the project presented with the synthetic jet. In another way, the heat transfer through a solid is always by conduction where the molecules of a solid remain constraint at relative positions. It is important to understand that convective heat transfer is complicated to analyse by the fact that it involves fluid motion. As well, the heat conduction of the plate gets hotter and cooler that chunks the fluid in contact, increasing the rates of conduction in a greater number of sites. Therefore, the convective heat transfer through fluid is much higher than by conduction. In the understanding, the convective heat in respect to the conductive is the value for the Nusselt number.

The Nusselt number is convection studies the non-dimensionalized governing equations and combine the variables. Different equations are applied to the model simulation to define the heat transfer coefficient or Nusselt number of the different factors that were mentioned before [5]. The heat transfer characteristics of low Reynolds number turbulent synthetic jets operating in a confined region while interacting with microchannel flow. The simulation consists of the definition of the convective heat transfer rate of a fluid dynamics of a laminar-boundary-layer flow system. We consider that it is at an incompressible steady flow the fluid field with constant viscosity, thermal conductivity, and specific heat. We will neglect the heat conduction in the direction of flow.

The Nusselt number consist of a fluid layer of thickness L and temperature difference $\Delta T = T_2 - T_1$, where the heat transfer through the fluid layer will be by convection when there is motion involved and on conduction heat transfer from the solid [6]. Heat flux which is the rate of heat transfer per unit time per unit surface will provide the conductive heat as:

Table 1: Heat transfer and fluid mechanics properties

$$q_{cov} = h\Delta T$$

$$q_{cond} = k\frac{\Delta T}{L}$$

$$\frac{q_{cov}}{q_{cond}} = \frac{h\Delta T}{k\frac{\Delta T}{L}}$$

$$Nu = \frac{q_{cov}}{q_{cond}} = \frac{h\mathrm{L}}{k}$$

And the heat transfer coefficient h is defined by:

$$\mathrm{h} = \frac{-k_{fluid}\left(\frac{\delta T}{\delta y}\right)_{y=0}}{T_w - T_\infty}$$

The Prandtl and Reynolds number is defined by:

$$Pr = \frac{\nu}{\alpha} = \frac{\mu/\rho}{k/c_p\rho} = \frac{c_p\mu}{k}$$

$$Re = \frac{\rho u x}{\mu}$$

Where:

q_{cov} - Convective heat transfer

q_{cond} - Conductive heat transfer

h - Heat transfer coefficient

k - Thermal conductivity of a fluid

$\left(\frac{\delta T}{\delta y}\right)_{y=0}$ - Temperature gradient

T_w - Wall temperature

T_∞ - Free-stream temperature

ν - Thickness of the hydrodynamics

α - Thermal boundary layers

c_p - Specific heat capacity

ρ - Fluid density

μ - Fluid viscosity

u - Uniform flow speed

x - Position over plate

IV. MODELING

The model consists of the following statistics:

Table 2: Geometric Statistics

Description	Value
Space dimension	2
Number of domains	1
Number of boundaries	12
Number of vertices	12

Table 2 present the information about the model geometric statistics. The vortices coordinates for the two different cases are presented in the following tables.

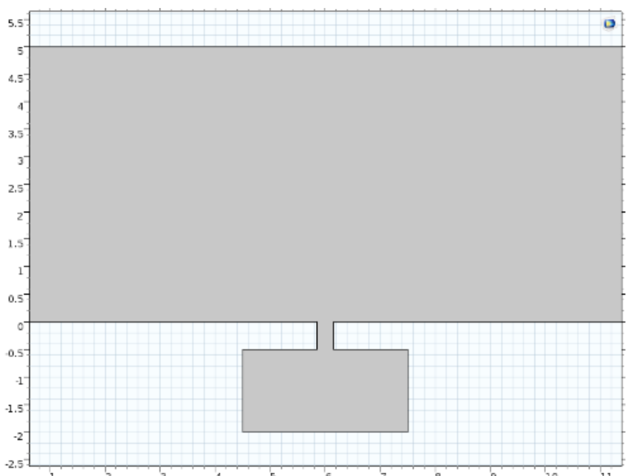
Table 3a: Symmetric geometry nodes coordinate

x (cm)	y (cm)
0	0
5.85	0
5.85	-0.5
4.5	-0.5
4.5	-2
7.5	-2
7.5	-0.5
6.15	-0.5
6.15	0
12	0
12	5
0	5
0	0

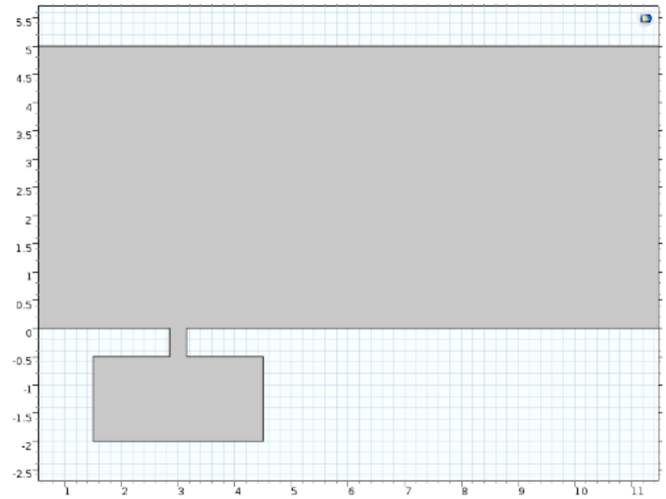
Table 3b: Geometry at a quarter of the total length nodes coordinates

x (cm)	y (cm)
0	0
2.85	0
2.85	-0.5
1.5	-0.5
1.5	-2
4.5	-2
4.5	-0.5
3.15	-0.5
3.15	0
12	0
12	5
0	5
0	0

The dimensions of the plate are random at the instance, yet the cavity and the airflow have a geometry with dimensions to simulate the moving flow. The dimensions are presented in Figure 3 and Table 3 presents the location of the nodes that model the design.



(a)



(b)

Figure 3: COMSOL solid geometry design based on nodes, a) COMSOL geometry symmetric structure, b) Geometry at a quarter of the length

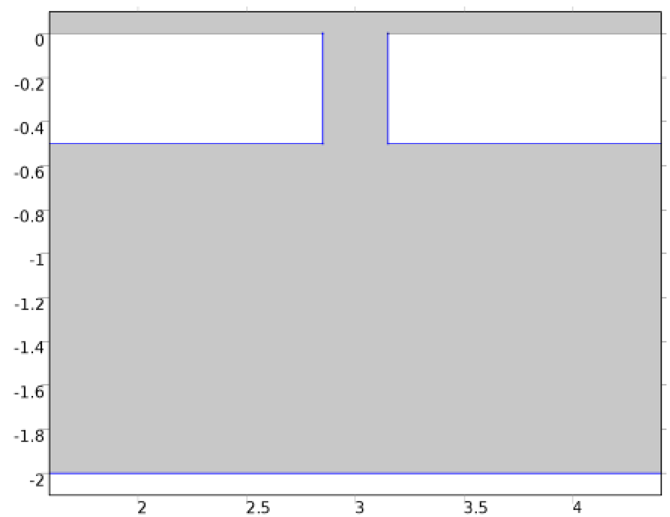


Figure 4: Thermal insulation of the module

The Figures 3a and 3b are the two location that the channel with the cavity will be located to understand the change in the heat transfer coefficient. In order to model the geometry, it is applied the polygon tool with the coordinates of the nodes. The coordinates are presented on the table 3a and 3b respectively with the design. Where the thermal insulation will be set at the channel and the cavity presented in figure 4.

V. PROCESS

The design and modelling of the experiments are conducted with COMSOL Multiphysics. The geometry consists in a two-dimensional system. The isolation of the wall is applied manually or the software defined the walls properties as presented in Figure 4. After, the cavity wall and diaphragm will be applied adiabatic conditions for the synthetic jet structure. Following the fluid mechanics condition is applied for the entire module as a fluid boundary with air as the material domain. The left boundary is set as the inlet a constant flow velocity. On the right boundary is set the outlet with zero pressure and the outflow is set on the same boundary. In the end, the heat transfer conditions are set on the surface with a heat flux over the two surfaces that are connecting to the channel. The walls over the fluid domain are defined at room temperature.

The parameters applied on all the cases are:

Table 4: Parameters

Name	Expression	Value	Description
k	0.0257[W/(m*K)]	0.0257 W/(m·K)	Thermal conductivity of the material
L	0.05[m]	0.050000 m	Characteristic length
Tw	293.15[K]	293.15 K	Temperature at the wall
Tinf	288[K]	288.00 K	Ambient temperature
Po	0[Pa]	0.0000 Pa	Pressure
q0	1000[W/m^2]	1000.0 W/m²	Heat flux
Uo	2[m/s]	2.0000 m/s	Uniform initial velocity

The parameters assist by providing an organized system for the application of heat transfer and fluid mechanics properties. The properties will assist to define the equations that will evaluate the Nusselt number and the heat transfer coefficient of the fluid.

As illustrated by (Jagannatha, Chandratilleke, and Narayanaswamy) [7], the paper presents that synthetic jets or pulsating jets lead to outstanding thermal performance in microchannel flow by increasing its heat dissipation rate by about 4.3 times compared to a channel with no synthetic jet. The experimental program applied to define this result was FLUENT for the computational fluid dynamics and to develop the solution domain for the figures that present the mesh generation was GAMBIT.

VI. Analytical Solution

The analytical solution will be computed on MATLAB and it will be presented the results on Table 6. The script applied for the computation of the local and average Nusselt number of the plate in a laminar flow system is based on the formulation presented in table 5.

Table 5: The local and average Nusselt number of the plate in a laminar flow system

Laminar flow, local is defined by: $q_w = cons, Re_x < 5 * 10^5, 0.6 < Pr < 50$	$Nu_x = 0.453\, {Re_x}^{1/2}\, Pr^{1/3}$
Also, when: $q_w = cons, Re_x < 5 * 10^5$	$Nu_x = \dfrac{0.4637\, {Re_x}^{1/2}\, Pr^{1/3}}{\left[1 + \left(\dfrac{0.0207}{Pr}\right)^{2/3}\right]^{1/4}}$
And in many cases the average Nusselt number in the Laminar flow is to be:	$Nu_L = 2\, Nu_{x=L} = 0.664\, {Re_L}^{1/2}\, Pr^{1/3}$

The values applied Prandtl number for air material is about:

$$Pr = 0.713$$

The value is obtained from the engineering toolbox website for the Air properties (20℃).

The MATLAB script will compute all these values for every case of speed;

```
% Local and Average Nusselt number
% Where:

Rho = input ('Fluid Density, Rho = ');
Mu = input ('Fluid Viscosity, Mu = ');
x = input ('Position over the Plate Length, x = ');
u = input ('Flow velocity, u = ');
% where the Reynolds number is defined by
Re= (Rho.*u.*x)./Mu;
Pr = input ('Prandtl number, Pr = ');
% To define the Local Nusselt number we have two formulas:
Nusl1= 0.453.*(Re.^(1./2).*Pr.^(1./3))
Nusl2=(0.4637.*(Re.^(1./2).*Pr.^(1./3)))./(1+(0.0207./Pr).^(2./
3)).^(1./4)
Nusa = 0.664.*(Re.^(1./2).*Pr.^(1./3))
```

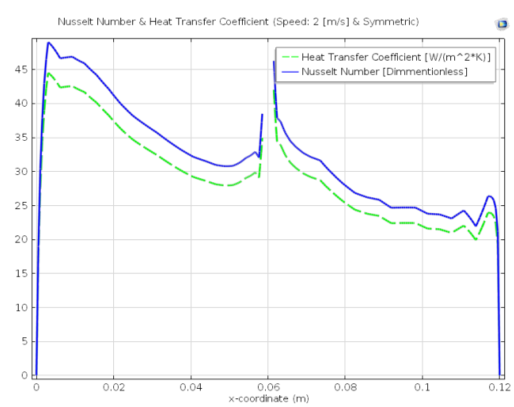
The results obtained from MATLAB applying the properties obtained for air come out to be:

Table 6: MATLAB results

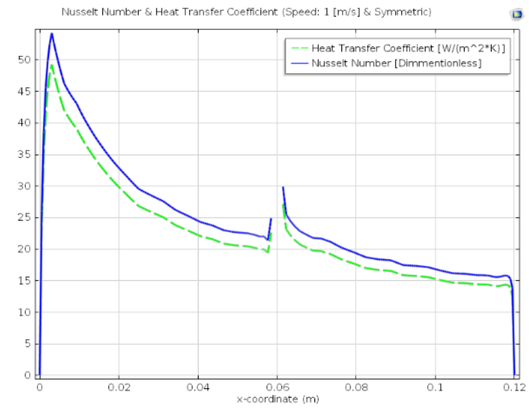
Fluid Density,	Fluid Density,
Rho = 1.205 $\left(\frac{kg}{m^3}\right)$	Rho = 1.205 $\left(\frac{kg}{m^3}\right)$
Fluid Viscocity,	Fluid Viscocity,
Mu = 4.626e-5 $\left(\frac{kg}{m*s}\right)$	Mu = 4.626e-5 $\left(\frac{kg}{m*s}\right)$
Position over the Plate Length, x = 0.12 (m)	Position over the Plate Length, x = 0.12 (m)
Flow velocity, u = 1 $\left(\frac{m}{s}\right)$	Flow velocity, u = 2 $\left(\frac{m}{s}\right)$
Prandtl number, Pr = 0.713	Prandtl number, Pr = 0.713
Nusl1 = 22.6261	Nusl1 = 31.9981
Nusl2 = 22.6437	Nusl2 = 32.0231
Nusa = 33.1649	Nusa = 46.9023

VII. RESULTS

The following figures present the local Nusselt number located over the entire laminar plater conducted in COMSOL. The solid lines presents the Nusselt number and the dotted lines presents the local heat transfer coefficient over the entire plate. Figures 5a and 5b present the change of the uniform flow velocity. Figures 6a and 6b present the change in uniform flow velocity with a change of the channel location.

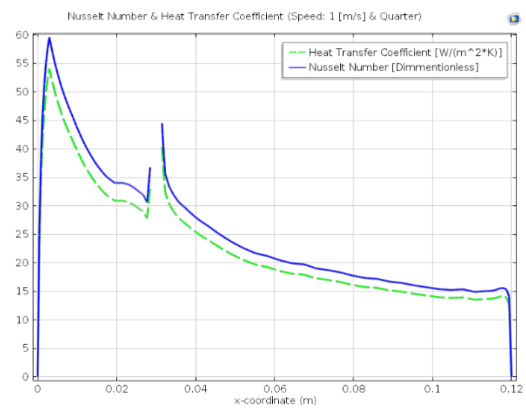


(a)

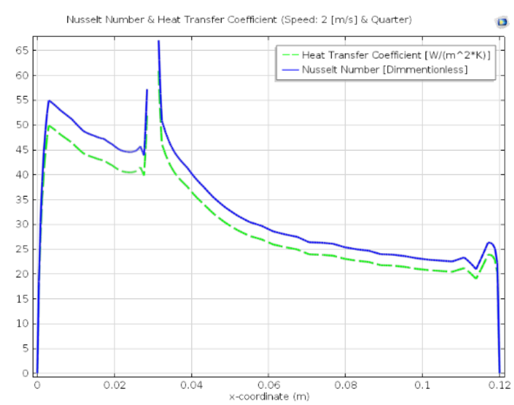


(b)

Figure 5: Average Nusselt number and heat transfer coefficient over the plate (Symmetric)



(a)



(b)

Figure 6: Average Nusselt number and heat transfer coefficient over the plate (Quarter)

The figures present the local Nusselt number plotted over the flat plate. The results present how the rate of heat diffused increases after the flow passes the channel for a period and after that it is reduce since the flow returns to be uniform.
The results collected from COMSOL of the average Nusselt number and heat transfer coefficient are presented in table 7:

Table 7: COMSOL average results

Model	**Heat Transfer Coefficient** [W/(m*K)]	**Nusselt number**
Symmetric [1 (m/s)]	22.415	24.586
Symmetric [2 (m/s)]	29.334	32.306
Quarter [1 (m/s)]	22.583	24.872
Quarter [2 (m/s)]	30.184	33.242

Table 8: Comparison of results

Model	**Nusselt Number (MATLAB**	**Nusselt number (COMSOL)**	**Percentage Difference** (%)
Symmetric [1 (m/s)]	22.644	24.586	7.8
Symmetric [2 (m/s)]	32.023	32.306	0.9

VIII. Discussion

The results present that the investigation in improving the heat dissipation with the conduction of the flow motion using synthetic jets can be improved by modifying the design. The heat transfer performance of single-phase flow on an electronics flat panel surface changes the Nusselt number by using a different uniform flow velocity and the position of the channel. The change in the frequency of the diaphragm that produces the synthetic jet from the channel will be conducted on the future work. Then, the results will explain how much it is the improvement of the heat dissipation with the synthetic jet system applied at different flow speed, the positioning of the channel and the synthetic jet from the change in the sinusoidal equation of the diaphragm at adiabatic conditions. At the moment the results present that the higher the flow, it will cool faster. This was mostly understood beforehand; since, if we consider our lungs as the synthetic jet system and we are blowing air at a hot surface. The higher the rate of the flow exerted from our lungs will cool faster the surface. However, the position of the channel influenced over the convective flow since it will produce some vortices after it pass over the channel. These vortices will increase the rate of the flow motion as turbulent flow. The change over the flow speed contributed in the heat dissipation. Figures 5 and 6 present the convection increment after the flow passes the channel. In order to confirm the values obtained from COMSOL are appropriated to be used. It was conducted the analytical procedure in MATLAB with the comparison of the COMSOL results of a thermal boundary layer. The average Nusselt number was defined from both software to be compared the differences in values. Table 8 presents how much is the percentage difference between both applications.

As it is presented in table 7, the case where the channel is located at one quarter from the initial length and the speed is double than the other case demonstrates that it is the best approach to define the most convenient heat dissipation system. After that the models set for the synthetic jet it will be possible to define how much it will influence the use of a synthetics jet over the same system. Therefore, the Nusselt number presents the enhancement of heat transfers through a fluid layer because of convection relative to conduction across the same fluid layer. The case that the averaged Nusselt number was larger is the case that the cavity was positioned to one-quarter of the plate and the flow velocity was greater. The results present that in this case, it is more effective the convection.

IX. Conclusion

Multiple CAD models are created to investigate the effect of the location, number of synthetic jets and the optimum operating frequency to improve the cooling process for heat generating electronics. Two standard cases of conventional convection cooling are simulated using COMSOL Multiphysics to compute the heat convection coefficient and Nusselt number. The results were close to the analytical solutions obtained from the MATLAB script presented. The results present that the procedure done on COMSOL can be considered acceptable for further tests. As expected, the increase of the incoming flow velocity improved the heat removed by convection. These solutions will be used as a benchmark for the synthetic jet cases for comparison of performance. The simulation of a synthetic jet is planned for the second phase of the present work.

X. Future work

The future work will involve extra simulation to evaluate the impact of the Nusselt number by changing the frequency of the diaphragm disc inside the cavity. In COMSOL, the approach is to apply a moving boundary at a time dependent study. However, it requires more research in order to stablish the correct approach to create the synthetic jet process over the laminar plate. It is also a desire to change the flow into a turbulent flow to define if the change will be different compared to the laminar flow results.

XI. WHAT COULD HAVE GONE WRONG

The results present a close approach with the analytical; however, the program computed the Nusselt number with the specified geometry that contains a channel on the layer. That might have changed the average value.

BIBLIOGRAPHY

[1] EDN Network. “What Causes Semiconductor Devices Fail?”, by V. Lakshminarayanan (November 1, 1999). http://www.edn.com/electronics-news/4382044/What-Causes-Semiconductor-Devices-to-Fail-.

[2] Smith, Barton L., and Ari Glezer. “The formation and evolution of synthetic jets”. *Physics of Fluids (1994-present)*, 10.9 (1998): 2281-2297.

[3] C. Revelo, A. Othamn, and V. Zulin, “Flow separation active control with synthetic jets actuators”. VC-JET 2014.

[4] Cengel, A. *Engineering and Mathematics*. New York: McGraw-Hill Science, 2006.

[5] Holman, J. P. *Heat transfer.* Boston: McGraw Hill Higher Education, 2010.

[6] Rohsenow, W. Hartnett, J. Cho, Y. *Handbook of Heat Transfer*. New York: McGraw-Hill Science, 1998.

[7] Taylor & Francis Group “Analysis of a synthetic jet – based electronic cooling module”, by Jagannatha, D. Narayanaswamy, R. Chandratilleke, T(2009). Numerical Heat Transfer, Part A: Applications 56 (3): 211-229. http://www.tandfonline.com/doi/abs/10.1080/10407780903163702#.VSvwpvnF_bI