COMPROMETIDA COM A PROMOÇÃO DO DESENVOLVIMENTO DA ENGENHARIA E DAS CIÊNCIAS MECÂNICAS
VI CONGRESSO NACIONAL DE ENGENHARIA MECÂNICA
VI NATIONAL CONGRESS OF MECHANICAL ENGINEERING
18 a 21 de agosto de 2010 – Campina Grande – Paraíba - Brasil
August 18 – 21, 2010 – Campina Grande – Paraíba – Brazil

STUDY OF INFLUENCE OF LAMINAR FLOW IN A RECTANGULAR CAVITY WITH THREE ELECTRONIC COMPONENTS

Valesca Alves Correa, valesca@unitau.br
José Rui Camargo, rui@unitau.br
Carlos Alberto Chaves, chaves@unitau.br
Luiz Eduardo Nicolini do Patrocínio, luiz@unitau.br

University of Taubaté, R. Daniel Danelli, s/n CEP: 12060-440,

Abstract: *The study of electronic components cooling is widely used in industrial development. In this paper we evaluated the influence of laminar flow in a rectangular cavity with three electronic components arranged in the cavity with several array of air-inlet steady in Cartesian coordinates with constant properties. There were used three-dimensional computer simulations with the software Ansys CFX, where the results take the best provisions to define the conditions for entry of air conditions. Special attention was paid to modeling the phenomenon, looking for increasing levels of complexity of the parameters speed and heat flux electronic components in order to monitor the effects of heat transfer. The results will be presented the thermal analysis of the influence of the arrangement of components in several arrangements in the cavity under mixed convection are investigating the best cooling conditions.*

Keywords: *mixed laminar channel flow, CFD, discrete heat sources, numerical investigation, electronic cooling.*

1. INTRODUCTION

The electronics industry is now developing at a fast rate and with it, the problems associated with the cooling of electronic packages is becoming increasing complex.

The electronics packaging design community is now commonly using CFD to thermally characterize the performance of electronic packages and systems. These simulation tools solve both fluid flow and temperature throughout the system. For fluid flow the classical Navier Stokes equations are solved.

Extensive documentation has been published which characterizes the flow structures present when studying fluid flow around a surface mounted cubical obstacle, whether referring to a single or matrix array of obstacles. The interested reader it is referred to the works of Meinders and Hanjalic (1999), who concentrate on low Reynolds number flow. Also the work of Martinuzzi and Tropea (1993) and Hussein and Martinuzzi (1996).

In many electronic cooling situations, arrays of heat-dissipating components are mounted on vertical (or inclined) parallel plate channels that are opened to the ambient at opposite ends. The simplest method of cooling these arrays is by circulating air vertically via natural convection. This method of electronic equipment cooling continues to play an important role in their thermal management, because it provides the advantage of low noise and high system reliability

A large number of studies on natural convection cooling of electronic components have been done in recent years. For example, Afrid and Zebib (1989) conducted a numerical study on single and multiple uniformly heated devices. They used a two-dimensional, conjugate laminar flow model and the analysis of their results lead to qualitative suggestions for improving the overall cooling of a multicomponent system. A similar configuration was considered by Said and Muhanna (1990), but the components were taken as protruding from one of the two vertical walls forming a channel. Sathe and Joshi ((1990) studied numerically heat dissipation by natural convection from a heat generating protrusion, mounted on a substrate inside a square enclosure.

Ramesh and Merzkirch (2001) present a study of steady, combined laminar natural convection and surface radiation from side-vented open cavities with top opening; Gunes and Liakopoulos (2003) study, by a spectral element method, the three-dimensional free convection in a vertical channel with spatially periodic, flush-mounted heat sources; Cheng and Lin (2005) present an optimization method of thermoelectric coolers using genetic algorithms and Vasiliev (2006) presents a short review on the micro and miniature heat pipes used as electronic component coolers. Devices applied for the cooling of electronic equipments are frequently based on forced convection (Sparrow et al., 1985). Altemani and Chaves (1988) present a numerical study of heat transfer inside a semi porous two-dimensional rectangular open cavity for both local and average Nusselt numbers at the heated wall and for the isotherms and streamlines of the fluid flowing inside the open cavity. Silva et al. (2007) present a work where it is done a numerical analysis of the heat transfer inside a semi porous two-dimensional rectangular open cavity, where forced and natural convection were considered.

Huang and Aggarwal (1995) investigated numerically the effects of wall conduction on cooling of a centred heat source in a two-dimensional rectangular enclosure. Heindel *et al.* (1995) performed two and three-dimensional calculations on laminar flow induced by a 3 x 3 array of discret heat sources flush-mounted to one vertical wall of a rectangular cavity whose opposite wall was isothermally cooled.

Flow separation and subsequent reattachment caused by sudden expansion in flow geometry, such as a backward-facing step, occurs in many engineering applications where heating or cooling is required. These applications appear in electronic cooling equipment, cooling of nuclear reactors, cooling of turbine blades, combustion chambers, environmental control systems, and many other heat transfer devices. A great deal of mixing of high and low energy fluid occurs in the separated and reattached flow regions, thus impacting significantly the heat transfer performance of these devices. Studies on separated flow have been conducted extensively and the backward-facing step geometry received most of the attention Simpson (1996).

The majority of published works on separated flow deal with the two-dimensional isothermal flow, and comparatively minority published about the heat transfer for the three-dimensional flow case. Iwai *et al.* (1999) reported on the forced convection results for a duct with an aspect ratio of 16.

The effects of step height on the three-dimensional flow and an array of discrete heat sources is the main objective of this study in order to give qualitative suggestions that may improve the thermal design of printed board assemblies.

2. GEOMETRY AND MODEL EQUATIONS

The three components composed of three discrete heat sources under study are mounted on a adiabatic square duct as illustrated in Fig. (1). All dimensions of the problem are 0.1m and are represented in Fig. (2). The region's air intlet was divided for computing simulations in nine sub-regions, where there were studied the different possibilities of grouping.

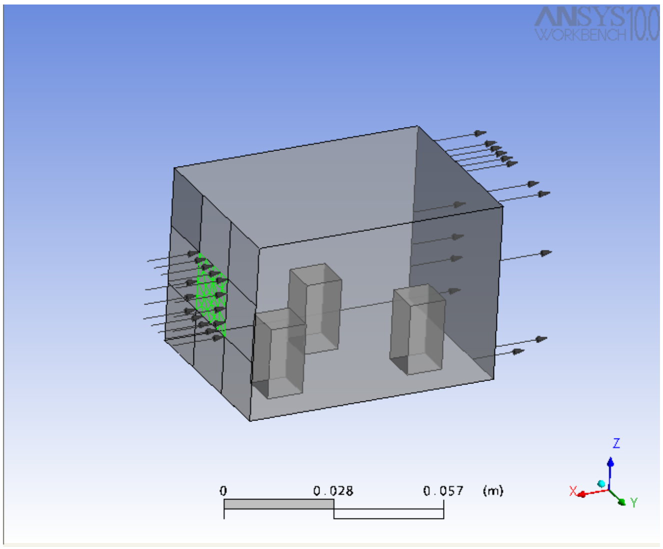


Figure 1. Coordinate system and thermal boundary conditions of the open cavity

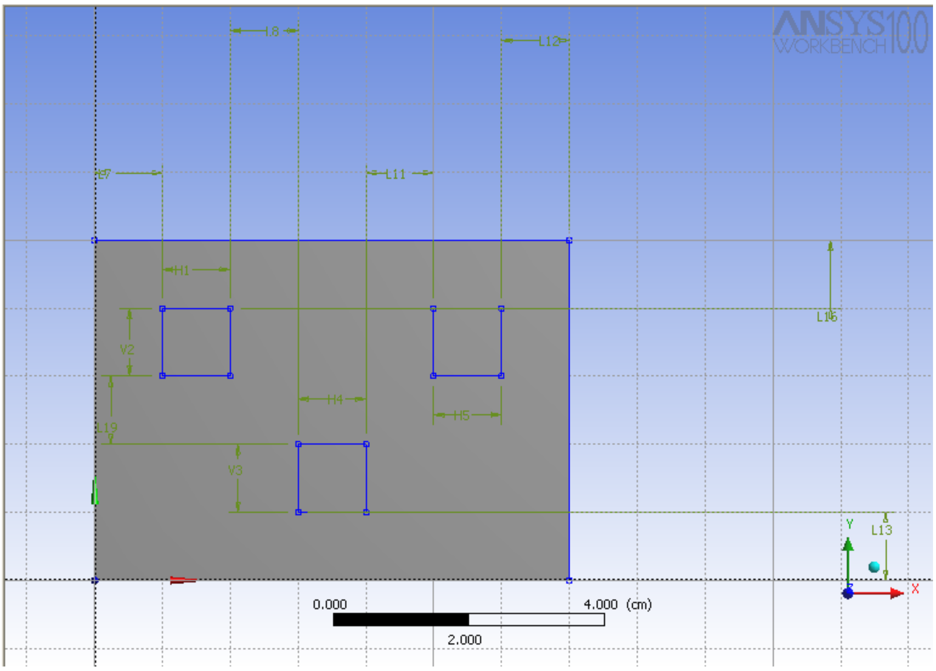


Figure 2. Coordinate system of the open cavity

It is assumed that the flow is laminar and occurs under steady state conditions. The laminar steady flow three-dimensional Navier-Stokes and energy equations with the continuity equation are expressed as follows.

Conservation of mass:

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

Conservation of momentum in x direction:

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

Conservation of momentum in y direction:

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

Conservation of momentum in z direction:

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

Conservation of energy:

$$\frac{\partial}{\partial x}(\rho \cdot C_p \cdot u \cdot T)+\frac{\partial}{\partial y}(\rho \cdot C_p \cdot v \cdot T)+\frac{\partial}{\partial z}(\rho \cdot C_p \cdot w \cdot T)=k \cdot \left(\frac{\partial^2 T}{\partial x^2}+\frac{\partial^2 T}{\partial y^2}+\frac{\partial^2 T}{\partial z^2}\right) \tag{5}$$

The physical properties are treated as constants and evaluated for air at the inlet temperature of T_0 = 20°C (i.e., ρ=1.205 kg/m^3, C_p= 1005 J/(kg °C), k = 0.0259 W/(m °C), μ= 1.81 x 10^{-5} kg/(m s) and Pr = 0.702).

Inlet flow is considered to be isothermal (T_0 = 20°C) with a constant distribution for the velocity component (u). The other velocity components (v) and (w) are set to be equal to zero at that inlet flow section. No slip boundary condition (zero velocities) is applied to all of the wall surfaces. Uniform and constant heat source of q = 1.000 W/m^2 is specified for the three heat sources, while the walls are treated as adiabatic surfaces. In this case the outlet air pressure was considered 1 atm as local atmospheric pressure.

3. NUMERICAL SOLUTION AND CODE VALIDATION

3.1. Numerical procedure

The system of coupled, non-linear, elliptic partial differential equation (1)-(5), subject to their boundary conditions, has been successfully solved by using the numerical method based on the "finite volume approach" with the aid of software ANSYS CFX (2008). This method, as others of the Simple family, is based on the spatial integration of conservation equations over finite volumes control. A second-order method has been used to compute the heat and momentum fluxes. The resulting "discretised equations" have been solved in a sequential manner, using the combination of the efficient multiple-and-alternate-sweeping technique, the "line-by-line" technique and the standard TDMA ("Three-Diagonal Matrix Algorithm"). On the other hand, the "pressure correction" equation, obtained by a combination of the discretised form of the Navier-Stokes equations and the continuity equation, has been employed not only to calculate the pressure field but also to correct the assumed velocity field during the calculation process in order to progressively satisfy all the discretised equations. Complete information regarding the numerical method is well documented elsewhere in Patankar (1980) and ANSYS CFX (2008).

The residuals resulting from the integration of the conservation equations were used as convergence indicators. During the iterative calculation process, these residuals were constantly monitored and scrutinized. For all the simulations performed in this study, converged solutions were usually achieved with residuals as low as 10^{-5} (or less) for all governing equations.

3.2. Discretisation grid

In order to ensure the accuracy of the results and their independence with respect to the number of nodes used in the discretisation process, several grids were tested and the 30,345 tetrahedrons with uniform grid has been found to be appropriate for the problem under study as showed on fig.(3). There were used elements layer inflation close to the solid surfaces for the three heat sources.

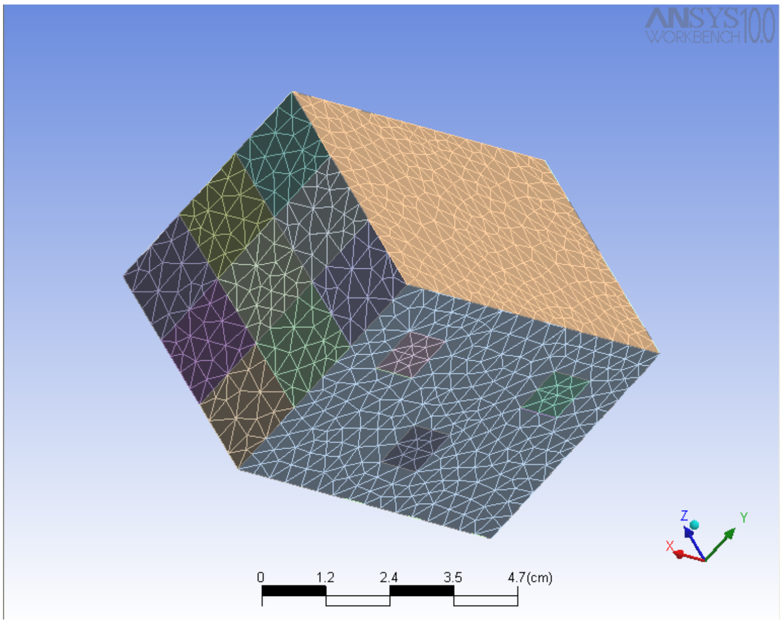
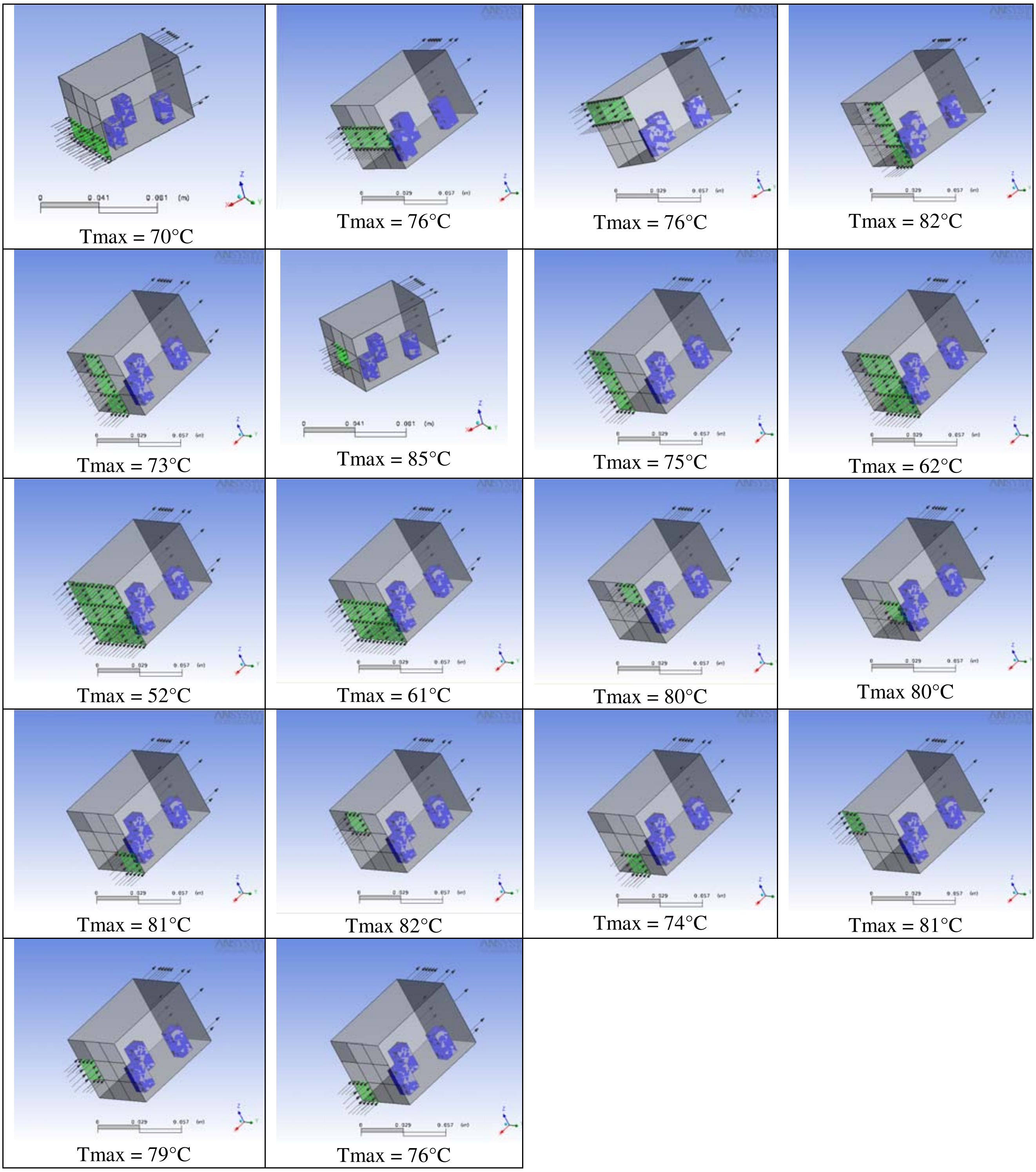


Figure 3. Mesh used in the geometry of electronic component

4. RESULTS AND DISCUSSION

The results presented in this paper have been calculated for values that corresponds to Inlet Velocity equal to 0.5 m/s and q = 1000 W/ m^2. Fig. (4) shows the severals possibilities simulated air into the electronic components with the corresponding maximum temperature (Tmax), where the best situation corresponds to the total group (Tmax = 52 ° C) and the worst case corresponds to a single air intake in the central region (Tmax = 85 ° C).



Tmax = 70°C

Tmax = 76°C

Tmax = 76°C

Tmax = 82°C

Tmax = 73°C

Tmax = 85°C

Tmax = 75°C

Tmax = 62°C

Tmax = 52°C

Tmax = 61°C

Tmax = 80°C

Tmax 80°C

Tmax = 81°C

Tmax 82°C

Tmax = 74°C

Tmax = 81°C

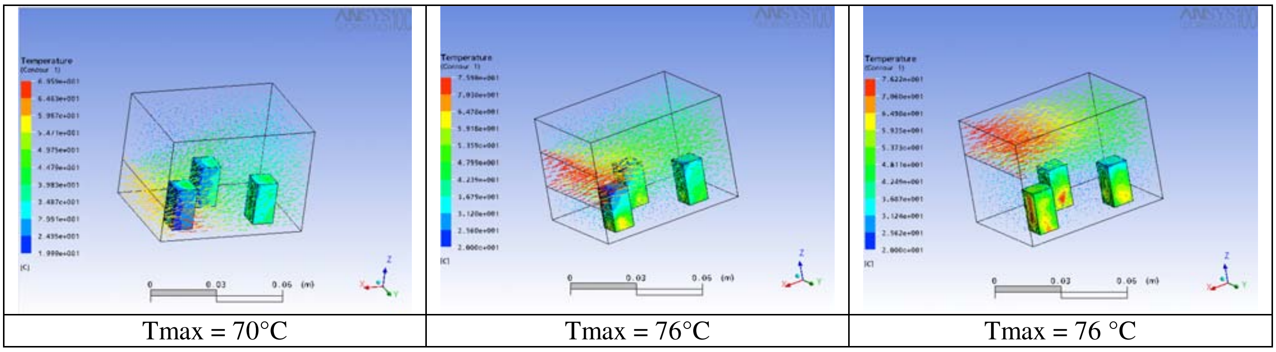
Tmax = 79°C

Tmax = 76°C

Figure 4. Schemes with severals simulated air inlets

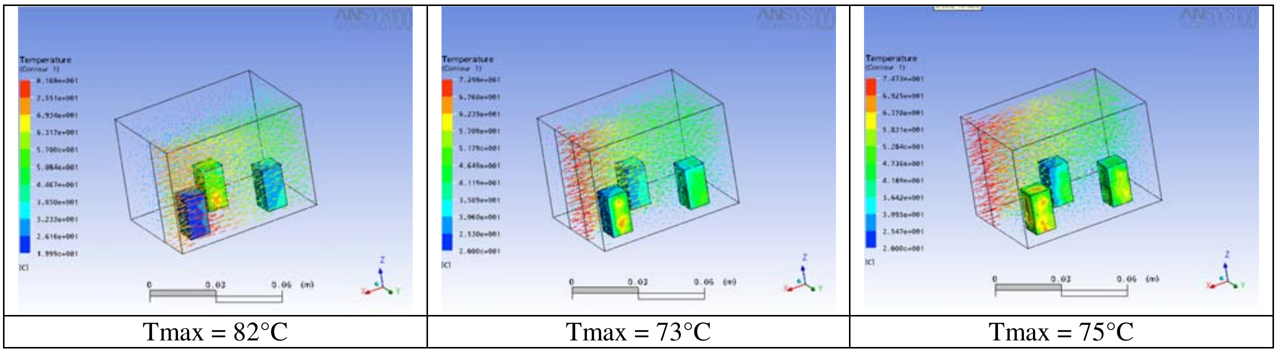
4.1. Evolution of velocity and temperature

Figures (5) to (11) show the severals arrangements in the simulated air inlet and the maximum temperatures obtained showing the velocity and temperature distribution of the three components.



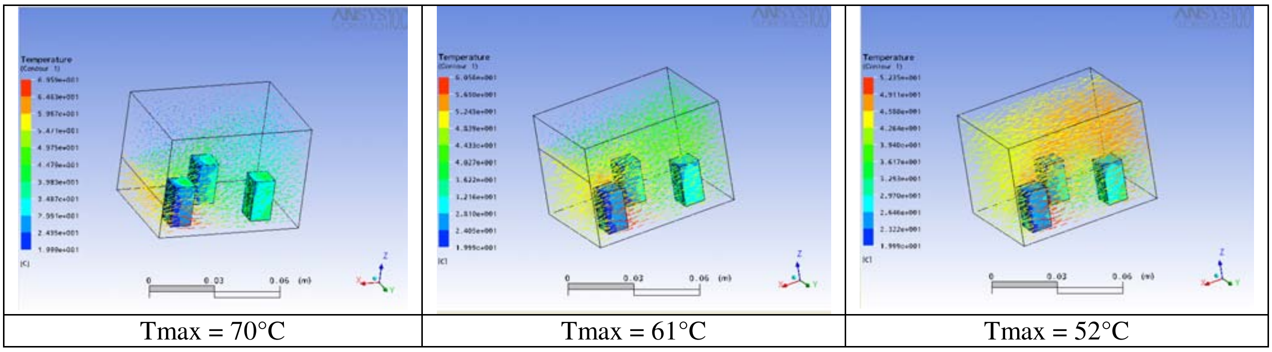
Tmax = 70°C | Tmax = 76°C | Tmax = 76 °C

Figure 5. Computer simulation with horizontal arrangement



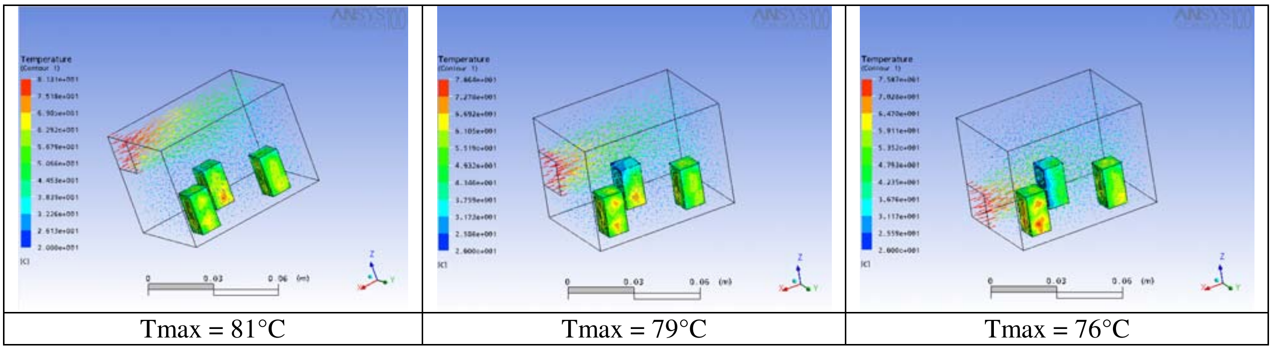
Tmax = 82°C | Tmax = 73°C | Tmax = 75°C

Figure 6. Computer simulation with vertical arrangement



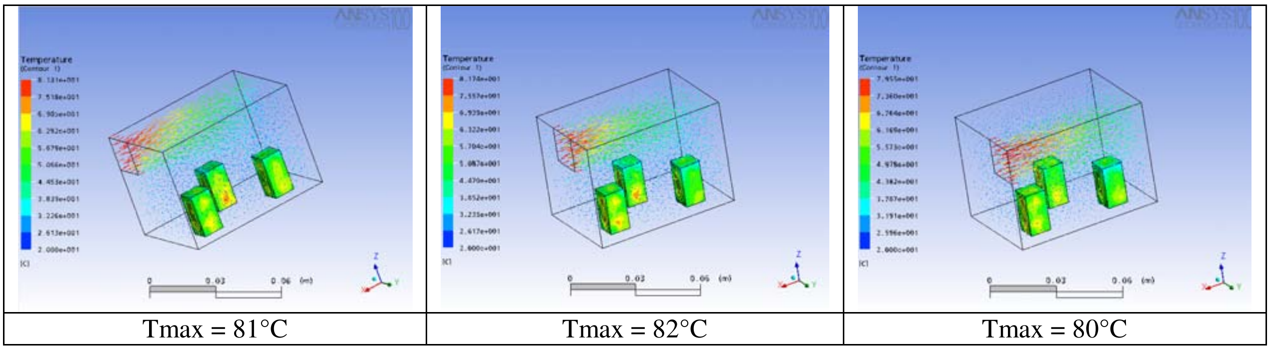
Tmax = 70°C | Tmax = 61°C | Tmax = 52°C

Figure 7. Computer simulation with horizontal arrangement with increased area



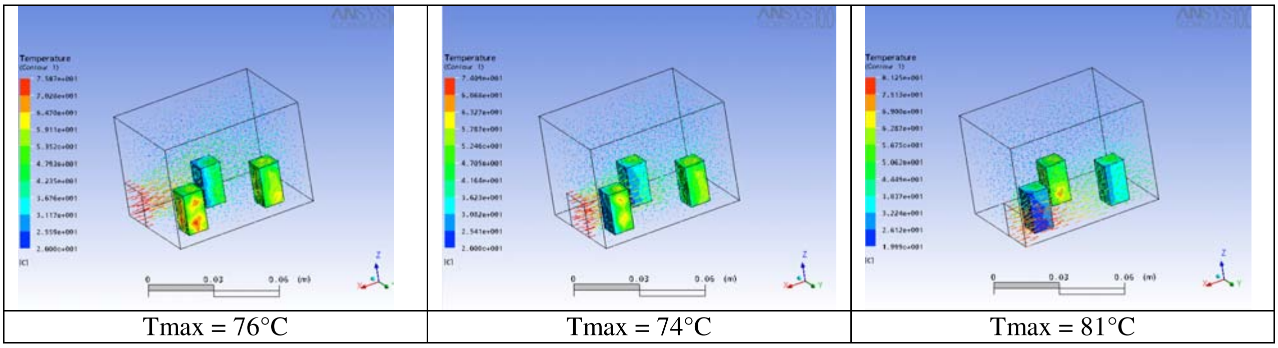
Tmax = 81°C | Tmax = 79°C | Tmax = 76°C

Figure 8. Computer simulation with a vertically disposed arrangement



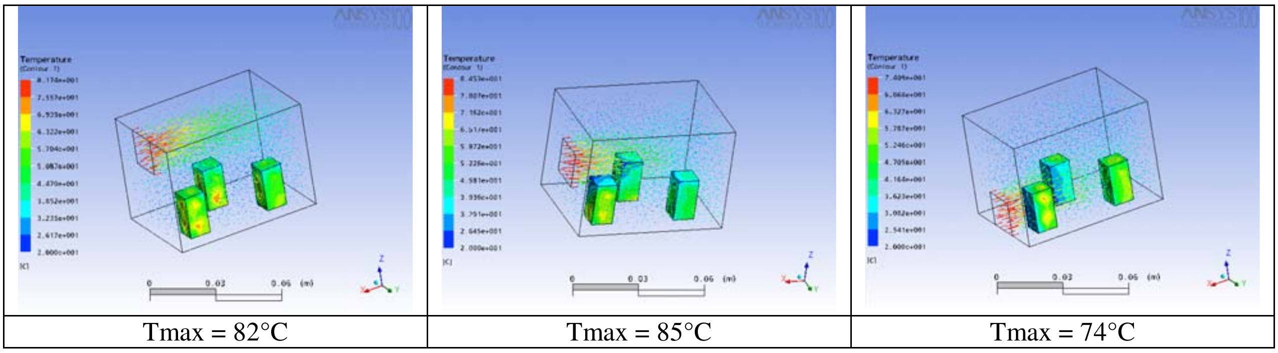
Tmax = 81°C | Tmax = 82°C | Tmax = 80°C

Figure 9. Computer simulation with single horizontal arrangement



Tmax = 76°C | Tmax = 74°C | Tmax = 81°C

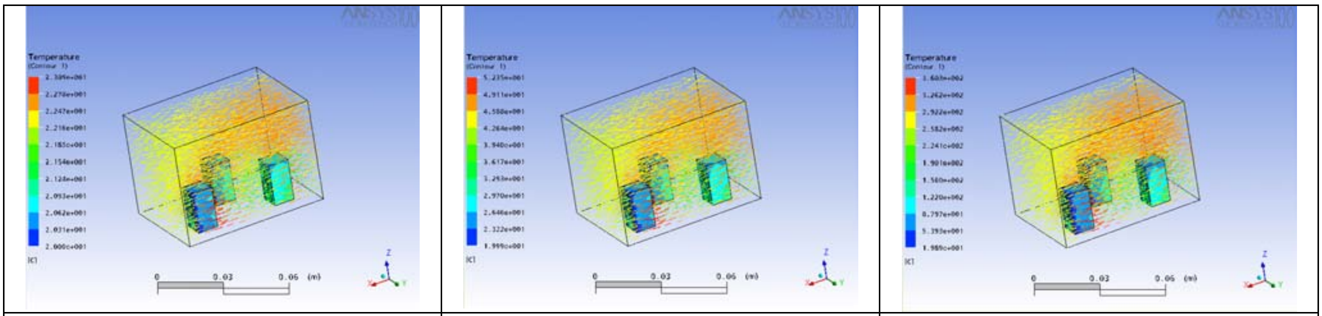
Figure 10. Computer simulation with single horizontal arrangement



Tmax = 82°C | Tmax = 85°C | Tmax = 74°C

Figure 11. Computer simulation with single vertical arrangement

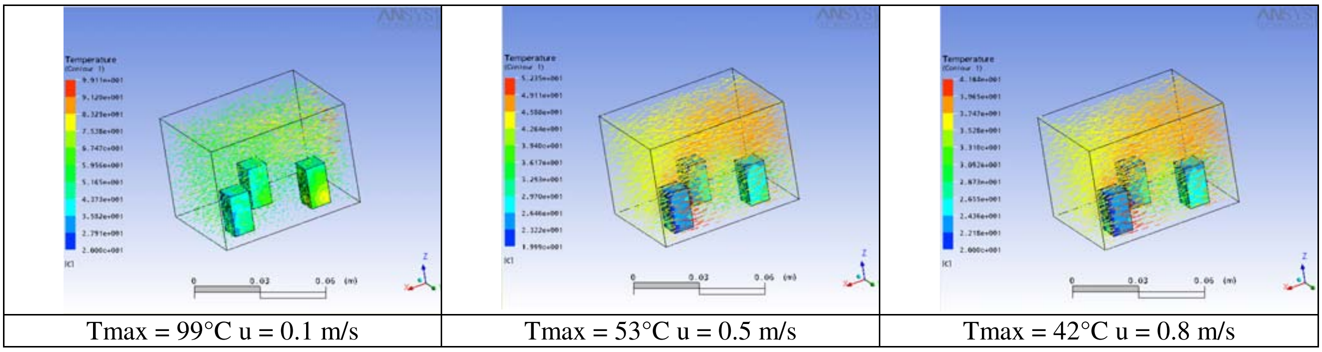
Figure (12) shows the simulations with a full array of inlet air, velocity of 0.5 m/s depending on the heat flow to the situations of 100, 1000 and 10000 W/m^2.



Tmax = 23°C q = 100 W/m2 | Tmax = 52°C q = 1000 W/m2 | Tmax = 360°C q = 10000 W/m2

Figure 12. Computer simulation of the total arrangement with speed of 0.5 m/s depending on the heat flux

Figure (13) shows the simulations with a full array of inlet air with heat flux of 1000 W/m^2 in terms of speed for the instances of 0.1, 0.5 and 0.8 m/s.



Tmax = 99°C u = 0.1 m/s | Tmax = 53°C u = 0.5 m/s | Tmax = 42°C u = 0.8 m/s

Figure 13. Computer simulation of the arrangement with the total heat flux 1000 W/m^2 depending on velocity

5. CONCLUSIONS

Laminar forced convection in a horizontal cavity has been studied by solving the corresponding partial equations numerically. The heat dissipation of electronic elements in a horizontal cavity is investigated by using computational fluid dynamics (CFD) methods in this paper. The velocity and temperature fields are simulated on the condition of several schemes with air as fluid. The effects of temperature and velocity field for the horizontal cavity have been analyzed by the position of inlet case. Then a cooling effects number have been put forward to evaluate the cooling effects of electronic elements. The optimized scheme of horizontal cavity has been obtained by comparing the cooling effect number of different schemes, which has established a solid foundation for discussing the cooling techniques of electronic elements further.

6. ACKNOWLEDGEMENTS

The authors thank the Universidade of Taubaté – UNITAU for their financial support to the present study.

7. REFERENCES

Altemani, C.A.C., Chaves, C.A., 1988, “Convective Cooling of a Semiporous Open Cavity”, XXVI National Heat Transfer Conference, Heat Transfer in Electronics – HTD: ASME, Philadelphia, vol. 111, pp. 149 -154.

ANSYS CFX, 2008. “CFX 10 User’s Guide.

Afrid, M. and Zebib, A., 1989, “Natural convection air cooling of heated components mounted on a vertical wall”, Numerical Heat Transfer, Part A 15, pp. 243-259.

Cheng, Y., Lin, W., 2005, “Geometric Optimization of Thermoelectric Coolers in a Confined Volume Using Genetic Algorithms”, Applied Thermal Engineering 25, pp. 2983-2997.

Gunes, H., Liakopoulos, A., 2003, “Three-dimensional Convective Cooling in a Vertical Channel with Flush-mounted Heat Sources”, Int. Journal of Heat and Mass Transfer 46, pp. 791-808.

Heindel, T.J., Ramadhyani, S. and Incropera, F. P., 1995, “Laminar natural convection in a discretely heated cavity: I-Asseessment of three-dimensional effects”, Journal of Heat Transfer, vol. 117, pp. 902-909.

Huang, Y. and Aggarwal, K., 1995, “Effect of wall conduction on natural convection in an enclosure with a centered heat source”, Journal of Electronic Packaging , vol. 117, pp. 301-306.

Hussein, H. J. and Martinuzzi, R. J., 1996, “Energy balance for turbulent flow around a surface mounted cube placed in a channel”, Physics of Fluids, vol. 8, pp. 764-780.

Iwai, H., Nakabe, K. and Suzuki, K., 1995, “Numerical simulation of buoyancy-assisting, backward-facing step flow and heat transfer in a rectangular duct”, Heat Transfer – Asian Res, vol. 28, pp. 58-76.

Martinuzzi, R. J. and Tropea, C., 1993, “The flow around surface-mounted, prismatic obstacles placed in a fully developed channel flow”, Journal of Fluid Engineering, vol. 115, pp. 85-92.

Meinders, E. R. and Hanjalic, K., 1999, “Vortex structure and heat transfer in turbulent flow over a wall-mounted matrix of cubes”, International Journal of Heat and Fluid Flow, vol. 20, pp. 255-267.

Patankar, S.V., 1980, “Numerical heat transfer and fluid flow”, Hemisphere Inc., McGraw-Hill, New York, NY, USA.

Ramesh, N., Merzkirch, W., 2001, “Combined Convective and Radiative Heat Transfer in Side-vented Open Cavities, Int. Journal of Heat and Fluid Flow, 22, pp. 180-187.

Said, S. A. and Muhanna, A., 1990, “Investigation of natural convection in a vertical parallel-walled channel with a single square obstruction”, in: A. F. Emery (Ed.), Simulation and Numerical Methods in Heat Transfer, vol. HTD 157, ASME, pp. 73-80.

Sathe, S. B. and Joshi, Y., 1990, “Natural convection liquid cooling of substrate-mounted protrusion in a square enclosure: effects of thermophysical properties, geometric dimensions and boundary conditions”, in: R. A. Wirtz, Gl L. Lehmann (Eds.), Thermal Modelling and Design of Electronic Systems and Device, vol. HTD 153, ASME, pp. 73-80.

Silva, M.N., Chaves, C.A., Altemani, C. A. C., Camargo, J. R., Corrêa, V. A., 2007 "Convective Cooling of Electronic Cooling", XIX Int. Congress of Mechanical Engineering", Brazil.

Sparrow, E.M., Stryker, P.C., Altemani, C.A.C., 1985 "Heat Transfer and Pressure Drop in Flow Passages that Open Along their Lateral Edges", Int. J. Heat Mass Transfer, vol. 28, n. 4, pp. 731-740.

Simpson, R.L., 1996, "Aspects of turbulent boundary-layer separation", Progress Aerosp. Sci., vol. 32, pp. 473-496.

Vasiliev, L. L., 2006, "Micro and Miniature Heat-pipes – Electronic component coolers", Applied Thermal Engineering, article in press.

8. RESPONSIBILITY NOTICE

The authors are the only responsible for the printed material included in this paper.