

TURBULENT MIXED FLOW APPLYING CFD IN ELECTRONIC COOLING

Carlos Alberto Chaves, carlos.chaves@unitau.br

Valesca Alves Corrêa, valesca@unitau.br

University of Taubaté – Mechanical Engineering Department
Rua Daniel Danelli s/n – Jd Morumbi – Taubaté – SP – Brazil

José Rui Camargo, rui@unitau.br

University of Taubaté – Mechanical Engineering Department
Rua Daniel Danelli s/n – Jd Morumbi – Taubaté – SP – Brazil

Luiz Eduardo Nicolini do Patrocinio Nunes, luiz@unitau.br

University of Taubaté – Mechanical Engineering Department
Rua Daniel Danelli s/n – Jd Morumbi – Taubaté – SP – Brazil

Vanderson Lopes, vanderson.lopes@br.atlascopco.com

University of Taubaté – Mechanical Engineering Department
Rua Daniel Danelli s/n – Jd Morumbi – Taubaté – SP – Brazil

***Abstract.** The thermal analysis of electronic equipment relies significantly on the use of Computational Fluid Dynamics (CFD) software to predict component heat dissipation. The commercial software CFX® was used in order to analyze the thermal parameters in electronic equipment where several values of velocity were explored in a model with a turbulent mixed forced cooling flow. The finite-volume method has been used to solve the problem equations throughout the physical domain with the standard $k-\epsilon$ turbulence model. To the numerical simulations there were analyzed the maximum temperature in function of the heat flux and injection velocity to laminar and turbulent flow and the visualization could also help with the selection of a numerical flow modeling strategy.*

***Keywords:** mixed turbulent flow, CFD, numerical analyses, electronic cooling*

1. INTRODUCTION

The heat transfer in enclosures has been studied for several engineering applications. Results have been presented in research surveys such as in Bruchberg et al. (1976), Kakaç et al. (1987), and it has become a main topic in convective heat transfer textbooks (Bejan, 1984). Usually the enclosures are closed and natural convection is the single heat transfer mechanism. There are however several applications in passive solar heating, energy conservation in building and cooling of electronic equipment, where open cavities are applied (Chan and Tien, 1985, Hess and Henze, 1984 and Penot, 1982). 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. Bessaih and Kadja (2000) described a numerical simulation of conjugate, turbulent natural convection air cooling of three heated components mounted on a vertical channel and showed the study of reduced temperatures. Silva et al. (2007) presented 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.

This paper presents a continued work (Silva et al., 2007) where it is done a turbulent numerical analysis of the heat transfer inside a semi porous two-dimensional rectangular open cavity. It is made by two vertical parallel plates opened at the top and closed at the bottom by a uniform heat flux, as indicated in Fig. 1. One of the vertical plates is porous and there is a normal forced fluid flow passing through. The opposite vertical plate supplies the same uniform heat flux to the cavity. In addition to the forced convection, the analysis considered the influence of natural convection effects. The maximum temperatures were obtained for the uniformly heated plate and to the bottom for the parameters like injection velocity and heat flux.

2. ANALYSIS

The conservation equations of mass, momentum and energy, as well as their boundary conditions, will be expressed for the system indicated in Fig. 1.

For an incompressible fluid, with constant thermo-physical properties, except for variation of density with temperature in the buoyancy force term (i.e. the Boussinesq approximation is valid), the governing equations for turbulent flow can be written in non-dimensional form as follow.

In order to obtain the conservation equations in dimensionless form, the following variables were defined:

Continuity equation

$$\frac{\partial U}{\partial X} + \frac{\partial V}{\partial Y} = 0 \quad (1)$$

Momentum equation in X direction

$$\frac{\partial(UU)}{\partial X} + \frac{\partial(VU)}{\partial Y} = -\frac{\partial P}{\partial X} + \text{Pr} \left\{ \frac{\partial}{\partial X} \left[(v^* + v_y^*) \frac{\partial U}{\partial X} \right] + \frac{\partial}{\partial Y} \left[(v^* + v_t^*) \frac{\partial U}{\partial Y} \right] \right\} \quad (2)$$

Momentum equation in Y direction

$$\frac{\partial(UV)}{\partial X} + \frac{\partial(VV)}{\partial Y} = -\frac{\partial P}{\partial Y} + \text{Pr} \left\{ \frac{\partial}{\partial X} \left[(v^* + v_y^*) \frac{\partial V}{\partial X} \right] + \frac{\partial}{\partial Y} \left[(v^* + v_t^*) \frac{\partial V}{\partial Y} \right] \right\} + Ra \cdot \text{Pr} \cdot \theta \quad (3)$$

Energy equation

$$\frac{\partial(U\theta)}{\partial X} + \frac{\partial(V\theta)}{\partial Y} = \left\{ \frac{\partial}{\partial X} \left[(k^* + \alpha_y^*) \frac{\partial \theta}{\partial X} \right] + \frac{\partial}{\partial Y} \left[(k^* + \alpha_t^*) \frac{\partial \theta}{\partial Y} \right] \right\} + S \quad (4)$$

Turbulent kinetic energy equation

$$\begin{aligned} \frac{\partial(UK)}{\partial X} + \frac{\partial(VK)}{\partial Y} = & \text{Pr} \left\{ \frac{\partial}{\partial X} \left[\left(v^* + \frac{v_t^*}{\sigma_k} \right) \frac{\partial K}{\partial X} \right] + \frac{\partial}{\partial Y} \left[\left(v^* + \frac{v_t^*}{\sigma_k} \right) \frac{\partial K}{\partial Y} \right] \right\} + \\ & \text{Pr} v_t^* \left[\left(\frac{\partial U}{\partial Y} + \frac{\partial V}{\partial X} \right)^2 + 2 \left(\frac{\partial U}{\partial X} \right)^2 + 2 \left(\frac{\partial V}{\partial Y} \right)^2 \right] - \varepsilon - Ra \frac{\text{Pr}^2}{\text{Pr}_t} v_t^* \frac{\partial \theta}{\partial Y} \end{aligned} \quad (5)$$

Rate of turbulent kinetic energy dissipation equation

$$\begin{aligned} \frac{\partial(U\varepsilon)}{\partial X} + \frac{\partial(V\varepsilon)}{\partial Y} = & \text{Pr} \left\{ \frac{\partial}{\partial X} \left[\left(v^* + \frac{v_t^*}{\sigma_\varepsilon} \right) \frac{\partial \varepsilon}{\partial X} \right] + \frac{\partial}{\partial Y} \left[\left(v^* + \frac{v_t^*}{\sigma_\varepsilon} \right) \frac{\partial \varepsilon}{\partial Y} \right] \right\} + \\ & C1 \text{Pr} v_t^* \frac{\varepsilon}{K} \left[\left(\frac{\partial U}{\partial Y} + \frac{\partial V}{\partial X} \right)^2 + 2 \left(\frac{\partial U}{\partial X} \right)^2 + 2 \left(\frac{\partial V}{\partial Y} \right)^2 \right] - C2 \frac{\varepsilon^2}{K} - C_e Ra \frac{\text{Pr}^2}{\text{Pr}_t} v_t^* \frac{\varepsilon}{K} \frac{\partial \theta}{\partial Y} \end{aligned} \quad (6)$$

The scales used for the non-dimensionalization of length, time, velocity, pressure, temperature, kinetic energy and dissipation are: L , L^2/α , α/L , $\rho(\alpha/L)^2$, QL^2/k_s , $(\alpha/L)^2$, α^3/L^4 , respectively. Ra and Pr are the Rayleigh and Prandtl numbers, defined respectively as:

$$Ra = \frac{g\beta[QL^2/k_s]}{\alpha\nu} \quad (7)$$

$$\text{Pr} = \frac{\nu}{\alpha} \quad (8)$$

where Q is the heat density, k_s the thermal conductivity of heated components, g the magnitude of gravitational acceleration, β the thermal expansion coefficient of the fluid, ρ its density, ν the kinematics viscosity and α the thermal diffusivity. The non-dimensional viscosity ν^* is ν/ν_{air} and is equal to 1 in the fluid region and ∞ within the solid regions corresponding to components. The non-dimensional turbulent eddy viscosity V_t^* is $(C_\mu/Pr) \cdot K^2/\mathcal{E}$ in the fluid domain and zero within the solid regions. The non-dimensional thermal conductivity is $k^* = k/k_{\text{air}}$, the non-dimensional turbulent thermal diffusivity α_t^* is $(Pr/Pr_t) \cdot V_t^*$.

The constants of the standard k - ϵ model are those given by Jones and Launder (1981) and are: $C_1=1.44$; $C_2=1.92$; $C_\mu=0.09$; $\sigma_\epsilon=1.00$; $\sigma_k=1.30$; and $Pr_t=1.00$. The constant C_e in the buoyant term of the dissipation equation was chosen as 0.70.

At the three solid boundaries of the open cavity, the velocity components are null, except the velocity of injection of the fluid (U_p) at the porous wall. The thermal boundary conditions comprise an uniform (reference) temperature at the porous wall and an specified heat flux at the heated vertical wall and in the bottom. Expressed in dimensionless terms, the boundary conditions become:

$$X=0 ; U_p = u_p \frac{D}{\nu} = Re_p ; V = 0 , \theta = 0 \quad (9a)$$

$$X=1 ; U=0 ; V=0 , \frac{\partial \theta}{\partial X}=1 \quad (9b)$$

$$Y=0 ; U=0 ; V = 0 , \frac{\partial \theta}{\partial Y}=1 \quad (9c)$$

where Re_p is the porous wall Reynolds number.

The equations of the turbulent kinetic energy and its dissipation, (5) and (6), were solved only in the fluid region. The turbulent kinetic energy is set to zero at solid walls, and its normal gradients are prescribed as zero at the other boundaries. The equation of dissipation is not solved at nodes that are adjacent to the wall.

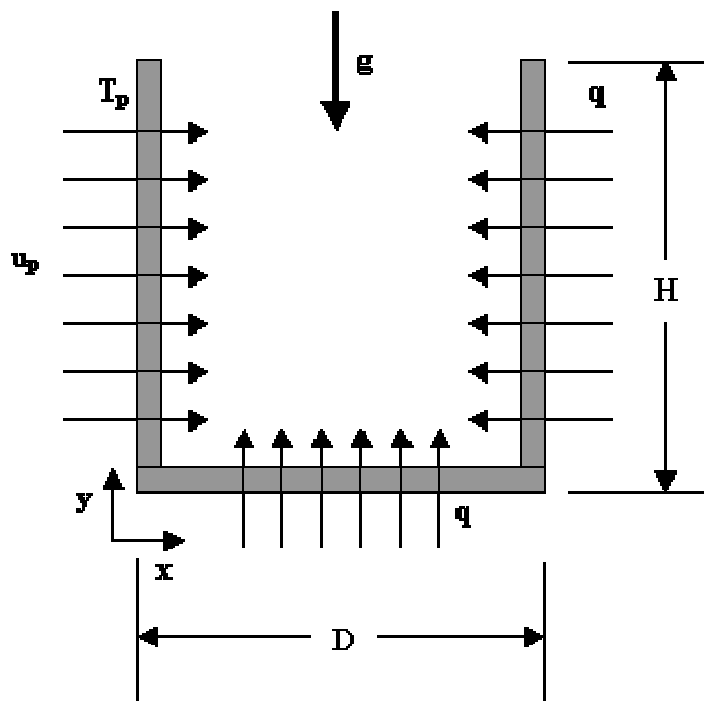


Figure 1. Coordinate system and thermal boundary conditions of the open cavity

3. NUMERICAL METHOD

The methodology consists in solving the physical problem using the finite volume CFD code CFX ® v. 10.0 from Ansys to discretize the two-dimensional partial differential equations of the mathematical model. Scalar quantities (P , θ ,

k and ϵ) are stored in the center of these volumes, whereas the horizontal and vertical components of the velocity (U and V) are stored on the faces. Power law profiles are used for the dependent variables to ensure realistic results.

Convergence of the numerical solution at consecutive time steps was reached when the mass, momentum and energy residuals was chosen after performing grid independency tests. Fig. 2 shows the mesh used in the representation of electronic component with 12062 nodes and 63984 tetrahedral elements.

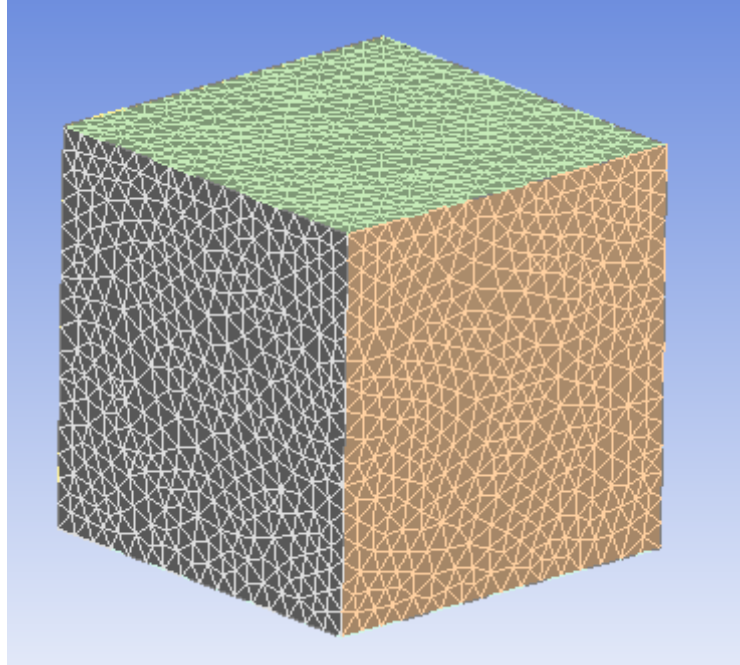


Figure 2. Mesh used in the geometry of electronic component

Fig. 3 shows the conditions defined in the geometry during the simulation realized in commercial software CFX® connected to Fig. 1 and the conditions expressed in Eq. 7 and 9.

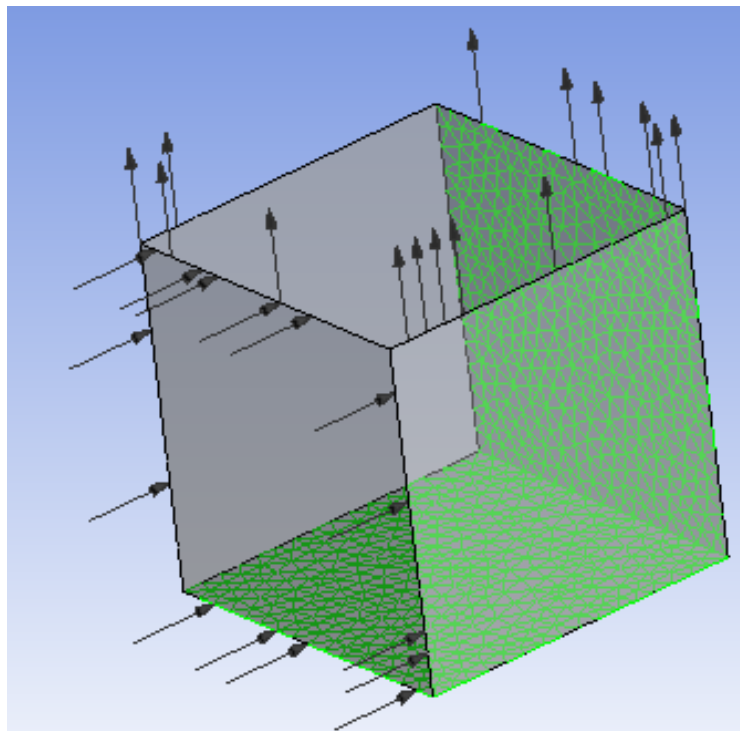


Figure 3. Representation of boundary conditions related to Fig. 1

Fig. 4 shows the evolution of convergence in parameters of momentum and mass to one of the simulations realized with injection velocity of 1 m/s and heat flux of 1 W/m². The decreasing behavior of the residues shows the condition of objection of realistic solutions and the error is under 10⁻³.

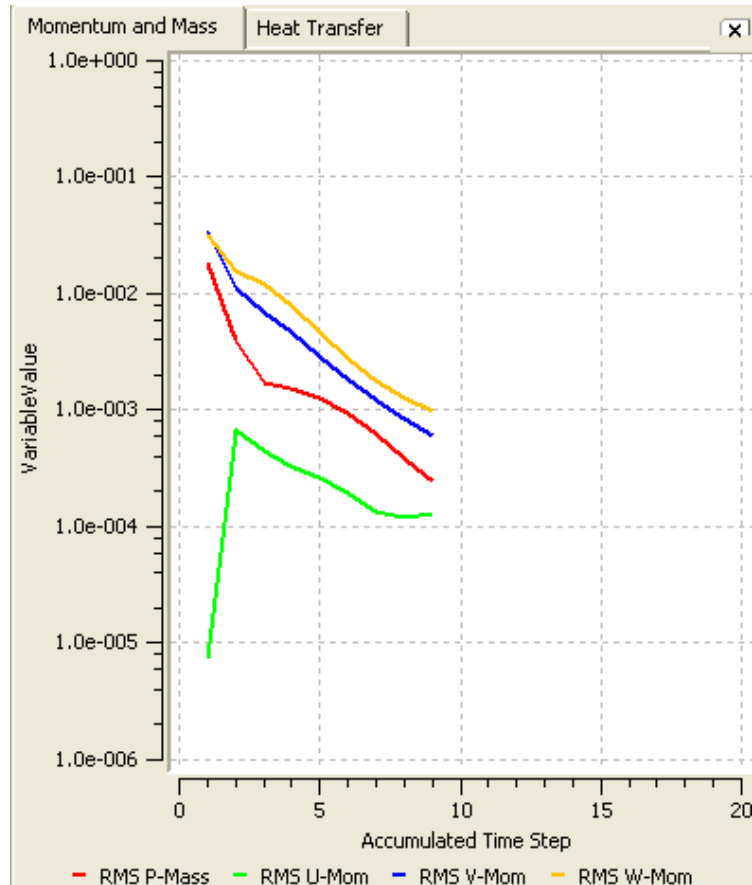


Figure 4. Representation of convergence in the parameters of momentum and mass

4. ANALYSIS OF RESULTS

The computer used had 1 GByte of RAM, considering a goal error equal to 10⁻³ for mass flow, velocity, turbulence and turbulent dissipation energy in CFX® v. 10.0.

For the simulation of cooling process it was used an electronic cooling with dimensions expressed in Tab 1.

Table 1. Values used to computational simulation.

Propriety	Value
width	0,1 [m]
height	0,1 [m]
depth	1 [m]
Air inlet temperature	25 [°C]
Atmospheric pressure	101,32 kPa

The parameters were applied in the commercial software CFX®, that executed the numerical simulation process of the air flow through the electronic equipment specified in Fig. 1, where the figures above express the results of maximum temperature inside cavity in function of injection velocity Up and heat flux parameters in the surface.

Figure (5) shows the effect of heat flux in the behavior of maximum temperature inside dominium studied to laminar and turbulent flows. It was observed the maximum temperature independence in relation to heat flux, where it variable with injection velocity Up.

From this value it was verified an intensity of maximum temperature depending on injection velocity Up to laminar and turbulent fluxes.

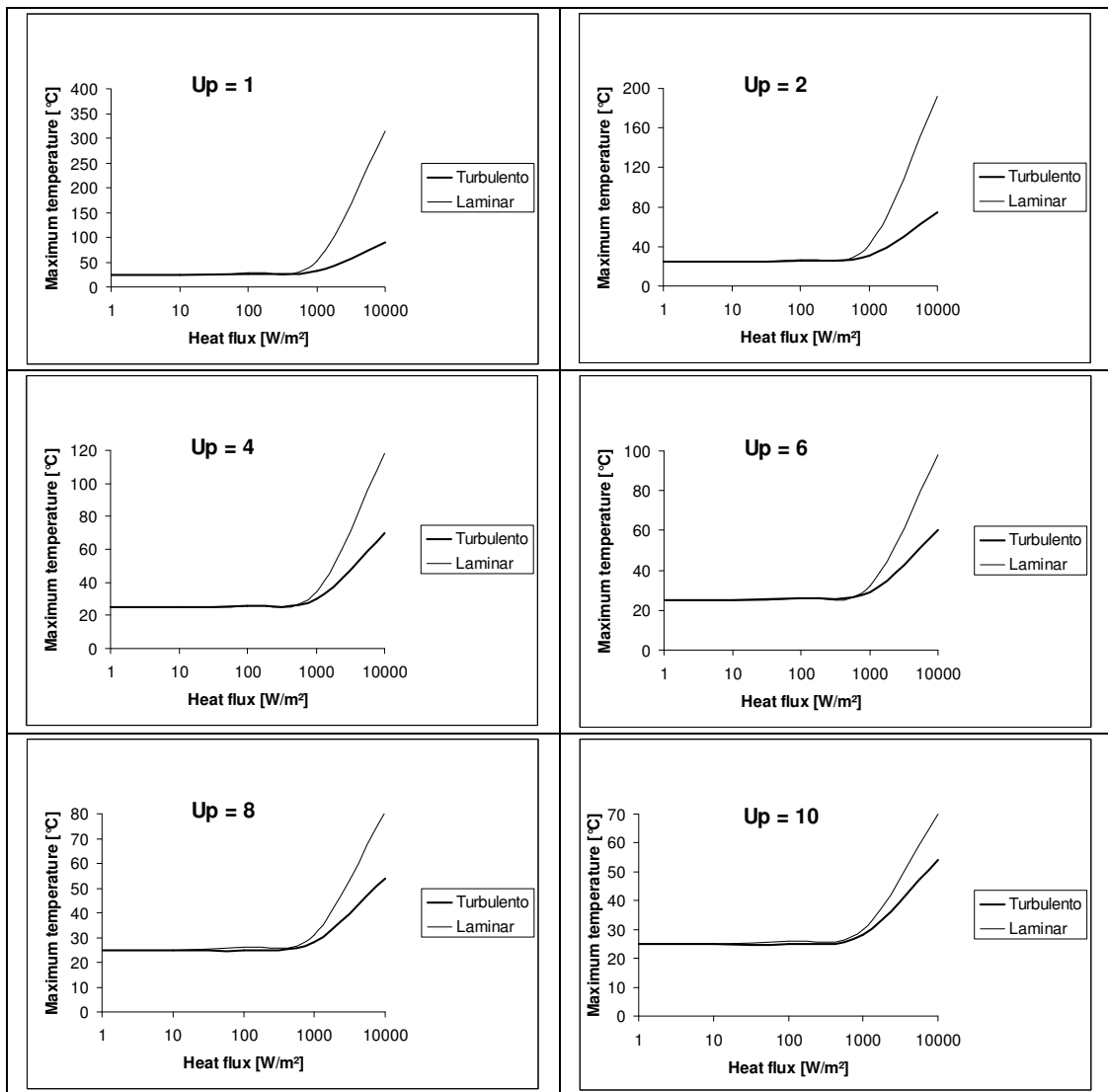


Figure 5. Laminar and turbulent representation to heat flux and maximum temperature to velocity U_p to 1, 2, 4, 6, 8 and 10

Figure (6) presents the behavior of maximum temperature inside studied domain in function of injection velocity U_p to laminar and turbulent fluxes to values of heat flux of 100, 1,000 and 10,000. It was observed the intensity influence of the kind of flux (laminar or turbulent) to the maximum temperature values.

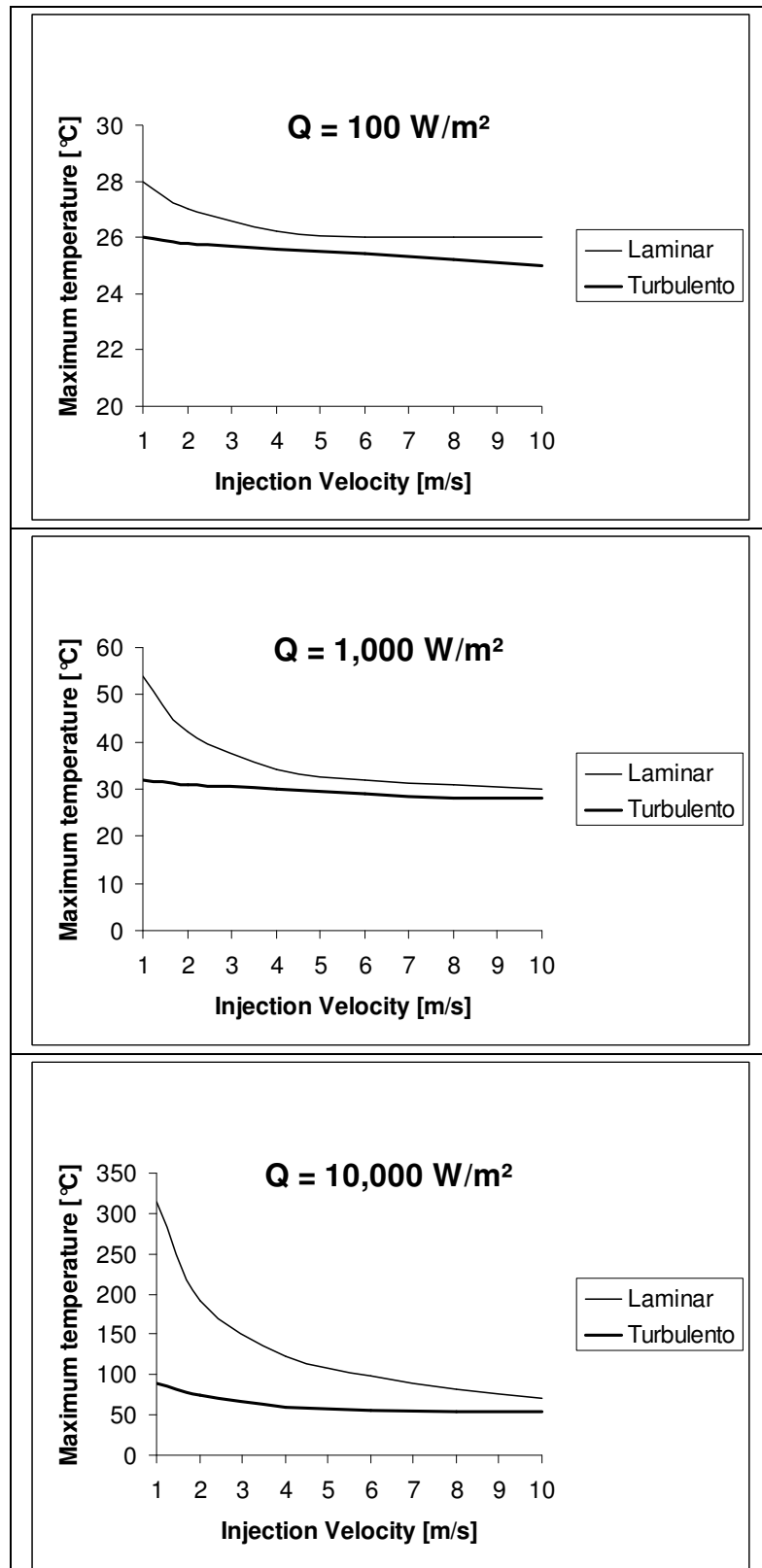


Figure 6. Laminar and turbulent representation to velocity and maximum temperature to heat flux of 100, 1,000 and 10,000

5. CONCLUSIONS

The mathematical model development was applied with techniques of computational fluid dynamic (CFD) and a cooling process with specified dimensions and defined geometry. The simulations of this process were realized with the

commercial software CFX® to evaluate the temperature distribution in function of air flux through an electronic component inside the cavity to laminar and turbulent fluxes. To all parameters considered, the turbulent flow showed the strongly reduced value of maximum temperature in the cavity. It was observed the maximum temperature independence in relation to injection velocity in the laminar and turbulent fluxes. It was also observed the continuum decrease of the temperature in relation to the velocity increase in the range of heat flux analyzed to the same conditions of laminar and turbulent fluxes.

The CFD methodology presented is a qualify application to analyze the cooling in electronic equipments.

6. 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.
- Bessaih, R., Kadja, M., "Turbulent Natural Convection Cooling of Electronic Components Mounted on a Vertical Channel", Applied Thermal Engineering, vol. 20, 2000, pp. 141 - 154.
- Bruchberg, H., Catton, I. Edwards, D.K., 1976, "Natural Convection in Enclosed Spaces - A Review of Applications to Solar Energy Collection". ASME J. of Heat Transfer, vol. 98, 1984, pp. 182 -188.
- Chan, Y. L., Tien, C.L., "A Numerical Study of Two-dimensional Laminar Natural Convection in Shallow Open Cavities", Int. J. Heat Mass Transfer, vol. 28, 1985, pp. 603 - 612.
- Cheng, Y., Lin, W., 2005, "Geometric Optimization of Thermoelectric Coolers in a Confined Volume Using Genetic Algorithms", Applied Thermal Engineering 25, pp. 2983-2997.
- Gune, 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.
- Hess, C.F., Henze, R.H., 1984, "Experimental Investigation of Natural Convection Losses from Open Cavities", ASME J. of Heat Transfer, vol. 106, pp. 33 - 338.
- Jones, W.P., Launder, B.E., 1981, "The Calculation of Low-Reynolds Number Phenomena with a Two-Equation Model of Turbulence", Int. Journal Heat Mass Transfer, vol. 16, pp. 1119 - 1130.
- Kakaç, S., Shah, R.K., Aung, W., 1987, "Handbook of Single Phase Convective Heat Transfer", J. Wiley.
- Patankar, S.V., 1980, "Numerical Heat Transfer and Fluid Flow", Mc Graw-Hill – Hemisphere.
- Penot, F., 1982, "Numerical Calculation of Two-dimensional Natural Convection in Isothermal Open Cavities", Numerical Heat Transfer, vol. 5, pp. 421 – 437.
- 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.
- 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.
- Silva, M.N. et al., 2007 "Convective Cooling of Electronic Cooling", XIX Int. Congress of Mechanical Engineering", Brazil.
- Vasiliev, L. L., 2006, "Micro and Miniature Heat-pipes – Electronic component coolers", Applied Thermal Engineering, article in press.

5. RESPONSIBILITY NOTICE

The author(s) is (are) the only responsible for the printed material included in this paper.