A NUMERICAL STUDY OF TURBULENT BOUNDARY LAYER FLOWS GENERATED BY NATURAL CONVECTION

Rafael Gabler Gontijo, rafaelgabler@yahoo.com.br
José Luiz Alves da Fontoura Rodrigues, fontoura@unb.br
Universidade de Brasília, Departamento de Engenharia Mecânica, 70910-900, Brasília - DF

Abstract. *The temperature and velocity fields, for two turbulent flows generated by natural convection were investigated by numerical simulation. The analyzed flows are the natural convection provided by a heated vertical flat plate and the natural convection inside a tall cavity with side walls in different temperatures. The numerical algorithm applies a consolidate Reynolds and Favre averaging process for the turbulent variables, and uses the classical $\kappa - \varepsilon$ model. The turbulent inner layer is modeled by one velocity and one temperature wall law. Spacial discretization is done by P1/isoP2 finite element method and temporal discretization is implemented using a semi-implicit sequential scheme of finite difference. The pressure-velocity coupling is numerically solved by a variation of Uzawa's algorithm. To filter the numerical noises, originated by the symmetric treatment to the convective fluxes, it is adopted a balance dissipation method. The remaining non-linearities, due to the laws of the wall, are treated by a minimal residual method. For the heated vertical flat plate 13179 nodes and 25840 elements were used, while for the tall cavity, 16317 nodes and 31680 elements were used in the isoP2 mesh. The numerical results are compared to experimental data are velocity and temperature profiles, and the Nusselt number is taken along the vertical wall.*

Keywords: *turbulence, natural convection, wall laws, $\kappa - \varepsilon$, finite element*

1. INTRODUCTION

Processes involving natural convection occurs in many aplications of industrial and enviromental interests. Many climatization systems are based in the natural convection phenomena provided by heated or cooled flat plates. In nature, many important processes are prevailed by natural convection, wich it's the main responsible for the ocean and the atmosphere motion as well as the heat transfer processes involved.

In most of the cases the flows generated by these bouyancy forces are turbulent, or have a predominant region of a turbulent regime. The interest in studing turbulent flows generated by natural convection, is not only industrial, but also scientific, since the lack of appropriate correlations to quatify the heat flux in the turbulent boundary layer compels us to appeal to experimental database, wich is an expensive way to obtain data. The numerical modeling, by the other hand, seams to be a cheaper way, but needs mindful validation by the comparison of the results obtained with an experimental database.

The main goal of this work is to provide an analysis of the methodology employed by a RANS algorithm in the modeling of the natural convection in turbulent boundary layer flows.

The physical reality to be numericaly modeled is based on the works of Nagano et. al (1992), and Betts and Bhokari (1996). The experimental work of Nagano studied the natural convection provided by a flat vertical plate of 4 meters height, wich was heated to the temperature of 60^{o}C in an ambient temperature of 15 ^{o}C. Temperature and velocity profiles were measured, and also the Nusselt number was taken along the wall. The reference velocity was of 0,0295 m/s, the profiles were taken at the heights of 1,92m; 2,54m; 3,24m.

The work done by Betts and Bhokari (1996), is the study of the natural convection yphenomena inside a tall cavity heated in one wall and cooled in the other one, so a recirculating flow is generated in it's interior. The cavity had 2,18 m height, 0,076 m of lenght, and 0,52 m wide. The flow is approximately bidimentional in the middle of the cavity, were the experimental database for comparison was taken.

The solver used, named Turbo2D, is a research Fortran numerical code, that has been continuously develloped by members of the Group of Complex Fluid Dynamics - Vortex, of the Mechanical Engineering Department of the University of Brasília, in the last twenty years. This solver is based on the adoption of the finite elements technique, under the formulation of pondered residuals proposed by Galerkin, adopting in the spatial discretization of the calculation domain the triangular elements of the type P1-isoP2, as proposed by Brison, Buffat, Jeandel and Serres (1985). The isoP2 mesh is obtained by dividing each element of the P1 mesh into four new elements. In the P1 mesh only the pressure field is calculated, while all the other variables are calculated in the isoP2 mesh.

Considering the uncertainties normally existing about the initial conditions of the problems that are numerically simulated, it is implemented a temporal integration of the governing equations system. In the pseudo transient process the initial state corresponds the beginning of the flow, and the final state occurs when the temporal variations of the velocity, pressure, temperature and other turbulent variables stop. The temporal discretization of the system of the governing equations, implemented by the algorithm of Brun (1988), uses sequential semi-implicit finite differences, with truncation error of order $0(\Delta t)$ and allows a linear handling of the equation system, at each time step.

The resolution of the coupled equations of continuity and momentum is done by a variant of Uzawat's algorithm proposed by Buffat (1981). The statistical formulation, responsible for the obtaining of the system of average equations, is done with the simultaneous usage of the Reynolds (1895) and Favre (1965) decomposition. The Reynolds stress of turbulent tensions is calculated by the $\kappa - \varepsilon$ model, proposed by Jones and Launder (1972) with the modifications introduced by Launder and Spalding (1974). The turbulent heat flux is modeled algebraically using the turbulent Prandl number with a constant value of 0,9.

In the program Turbo2D, the boundary conditions of velocity and temperature can be calculated by velocity and temperature wall laws. In this work, it is used the classic logarithm wall law for velocity and temperature. The numerical instability resulted of the explicit calculation of the boundary conditions of velocity, trough the evolutive temporal process, is controlled by the algorithm proposed by Fontoura Rodrigues (1990). The numerical oscillations induced by the Galerkin formulation, resulting of the centered discretization applied to a parabolic phenomenon, that is the modeled flow, are cushioned by the technique of balanced dissipation, proposed by Huges and Brooks (1979) and Kelly, Nakazawa and Zienkiewicz (1976) with the numerical algorithm proposed by Brun (1988).

In order to quantify the wideness of range and the consistence of the numerical modeling done by the solver Turbo2D, the wall heat fluxes obtained numerically are compared to the experimental data of Nagano et. al (1992), and the temperature and velocity profiles are compared to the datas of Betts and Bhokari (1996) and Nagano et. al (1992).

2. GOVERNING EQUATIONS

The system of non-dimensional governing equations, for a dilatable and one phase flow, without internal energy generation, and in a subsonic regime (Mach number under 0,3) is:

$$\frac{\partial \rho}{\partial t} + \frac{\partial (\rho u_i)}{\partial x_i} = 0, \tag{1}$$

$$\frac{\partial \rho u_i}{\partial t} + \frac{\partial}{\partial x_j}(\rho u_i u_j) = -\frac{\partial \underline{p}}{\partial x_i} + \frac{1}{Re}\frac{\partial}{\partial x_j}\left[\mu\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right)\right] - \frac{2}{3Re}\frac{\partial}{\partial x_j}\left(\mu \frac{\partial u_k}{\partial x_k}\delta_{ij}\right) + \frac{1}{Fr}\rho\frac{g_i}{\|g\|}, \tag{2}$$

$$\frac{\partial (\rho T)}{\partial t} + \frac{\partial (\rho u_i T)}{\partial x_i} = \frac{1}{RePr}\frac{\partial}{\partial x_i}\left(k\frac{\partial T}{\partial x_i}\right) \tag{3}$$

$$\rho(T+1) = 1. \tag{4}$$

In this system of equations ρ is the fluid density, t is the time, x_i are the space cartesian coordinates in tensor notation, μ is the dynamic viscosity coefficient, δ_{ij} is the Kronecker delta operator, g_i is the acceleration due to gravity, T is the absolute temperature, u_i is the flow velocity , k is the thermal conductivity, Re is the Reynolds number, Fr is the Froud number, Pr is the Prandtl number, and the non dimensional pressure is

$$\underline{p} = \frac{p - p_m}{\rho_o U_o^2} \tag{5}$$

where p_m is the average spatial value of the pressure field, p is the actual value of pressure, ρ_0 and u_0 are the reference values of the fluid density and the flow velocity. More details about the dimensionless process are given by Brun (1988). In order to simplify the notation adopted, the variables in their dimensionless form have the same representation as the dimensional variables. The Reynolds, Prandtl and Froude numbers are defined with the reference values adopted in this process.

2.1 THE TURBULENCE MODEL

In this work all the dependent variables of the fluid are treated as a time average value plus a fluctuation of this variable in a determinate point of space and time. In order to account variations of density, the model used applies the well known Reynolds (1985) decomposition to pressure and fluid density and the Favre (1965) decomposition to velocity and temperature. In the Favre (1965) decomposition a randomize generic variable φ is defined as:

$$\varphi(\vec{x},t) = \widetilde{\varphi}(\vec{x}) + \varphi''(\vec{x},t) \qquad \text{with} \qquad \widetilde{\varphi} = \frac{\overline{\rho\varphi}}{\bar{\rho}} \qquad \text{and} \qquad \overline{\varphi''}(\vec{x},t) \neq 0. \tag{6}$$

Applying the Reynolds (1895) and Favre (1965) decompositions, to the governing equations, and taking the time average value of those equations, we obtain the mean Reynolds equations:

$$\frac{\partial \overline{\rho}}{\partial t} + \frac{\partial}{\partial x_i}(\overline{\rho}\widetilde{u}_i) = 0, \tag{7}$$

$$\frac{\partial}{\partial t}\left(\overline{\rho}\widetilde{u}_i\right)+\frac{\partial}{\partial x_j}\left(\overline{\rho}\widetilde{u}_j\widetilde{u}_i\right)=-\frac{\partial\overline{p}}{\partial x_i}+\frac{\partial}{\partial x_j}\left[\overline{\tau_{ij}}-\overline{\rho u_j'' u_i''}\right]+\overline{\rho}g_i, \tag{8}$$

where the viscous stress tensor is

$$\overline{\tau_{ij}}=\mu\left[\left(\frac{\partial\widetilde{u}_i}{\partial x_j}+\frac{\partial\widetilde{u}_j}{\partial x_i}\right)-\frac{2}{3}\frac{\partial\widetilde{u}_l}{\partial x_l}\delta_{ij}\right], \tag{9}$$

$$\frac{\partial(\overline{\rho}\widetilde{T})}{\partial t}+\frac{\partial(\widetilde{u}_i\widetilde{T})}{\partial x_i}=\frac{\partial}{\partial x_i}\left(\alpha\frac{\partial\widetilde{T}}{\partial x_i}-\overline{\rho u_i'' T''}\right) \tag{10}$$

$$\overline{p}=\overline{\rho}R\widetilde{T} \tag{11}$$

In these equations α is the molecular thermal diffusivity and two news unknown quantities appear in the momentum (8) and in the energy equation (10), defined by the correlations between the velocity fluctuations, the so-called Reynolds Stress, given by the tensor $-\overline{\rho u_i'' u_j''}$, and by the fluctuations of temperature and velocity, the so-called turbulent heat flux, defined by the vector $-\overline{\rho u_i'' T''}$.

The Reynolds stress of turbulent tensions is calculated by the $\kappa-\varepsilon$ model. For flows with variable density, it is adopted the formulation of Jones and McGuirk (1979),where

$$-\overline{\rho u_i'' u_j''}=\mu_t\left(\frac{\partial\widetilde{u}_i}{\partial x_j}+\frac{\partial\widetilde{u}_j}{\partial x_i}\right)-\frac{2}{3}\left(\overline{\rho}\kappa+\mu_t\frac{\partial\widetilde{u}_l}{\partial x_l}\right)\delta_{ij}, \tag{12}$$

the turbulent kinetic energy is done by

$$\kappa=\frac{1}{2}\overline{u_i'' u_i''}. \tag{13}$$

and

$$\mu_t=C_\mu\bar{\rho}\frac{\kappa^2}{\varepsilon}=\frac{1}{Re_t}\ , \tag{14}$$

where ε is the rate of dissipation of the turbulent kinetic energy. The turbulent heat flux is modeled algebraically using the turbulent Prandl number Pr_t equal to a constant value of 0,9 by the relation

$$-\overline{\rho u_i'' T''}=\frac{\mu_t}{Pr_t}\frac{\partial\widetilde{T}}{\partial x_i}. \tag{15}$$

In the equation (14) C_μ is a constant of calibration of the model, that values $0,09$. Once that κ and ε are additional variables, we need to know there transport equations. The transport equations of κ and ε were deduced by Jones and Launder (1972), and the closed system of equations to the $\kappa-\varepsilon$ model is given by:

$$\frac{\partial\bar{\rho}}{\partial t}+\frac{\partial(\bar{\rho}\widetilde{u}_i)}{\partial x_i}=0\ , \tag{16}$$

$$\frac{\partial\left(\bar{\rho}\widetilde{u}_i\right)}{\partial t}+\widetilde{u}_j\frac{\partial\left(\bar{\rho}\widetilde{u}_i\right)}{\partial x_j}=-\frac{\partial\bar{p}^*}{\partial x_i}+\frac{\partial}{\partial x_j}\left[\left(\frac{1}{Re}+\frac{1}{Re_t}\right)\left(\frac{\partial\widetilde{u}_i}{\partial x_j}+\frac{\partial\widetilde{u}_j}{\partial x_i}\right)\right]+\frac{1}{Fr}\bar{\rho}g_i\ , \tag{17}$$

$$\frac{\partial\left(\bar{\rho}\widetilde{T}\right)}{\partial t}+\widetilde{u}_j\frac{\partial\left(\bar{\rho}\widetilde{T}\right)}{\partial x_j}=\frac{\partial}{\partial x_j}\left[\left(\frac{1}{Re\,Pr}+\frac{1}{Re_t\,Pr_t}\right)\frac{\partial\widetilde{T}}{\partial x_j}\right]\ , \tag{18}$$

$$\frac{\partial\left(\bar{\rho}\kappa\right)}{\partial t}+\widetilde{u}_i\frac{\partial\left(\bar{\rho}\kappa\right)}{\partial x_j}=\frac{\partial}{\partial x_i}\left[\left(\frac{1}{Re}+\frac{1}{Re_t\,\sigma_\kappa}\right)\frac{\partial\kappa}{\partial x_i}\right]+\Pi-\bar{\rho}\varepsilon+\frac{\bar{\rho}\beta g_i}{Re_t\,Pr_t}\frac{\partial\widetilde{T}}{\partial x_i}\ , \tag{19}$$

$$\begin{aligned}\frac{\partial\left(\bar{\rho}\varepsilon\right)}{\partial t}+\widetilde{u}_i\frac{\partial\left(\bar{\rho}\varepsilon\right)}{\partial x_j}&=\frac{\partial}{\partial x_i}\left[\left(\frac{1}{Re}+\frac{1}{Re_t\,\sigma_\varepsilon}\right)\frac{\partial\varepsilon}{\partial x_i}\right]\\&\quad+\frac{\varepsilon}{\kappa}\left(C_{\varepsilon1}\Pi-C_{\varepsilon2}\bar{\rho}\varepsilon+C_{\varepsilon3}\frac{\bar{\rho}\beta g_i}{Re_t\,Pr_t}\frac{\partial\widetilde{T}}{\partial x_i}\right)\ ,\end{aligned} \tag{20}$$

$$\bar{\rho}\left(1+\widetilde{T}\right)=1\ , \tag{21}$$

where:

$$\frac{1}{Re_t} = C_\mu \bar{\rho} \frac{\kappa^2}{\varepsilon} \ , \tag{22}$$

$$\Pi = \left[\left(\frac{1}{Re_t} \right) \left(\frac{\partial \widetilde{u}_i}{\partial x_j} + \frac{\partial \widetilde{u}_j}{\partial x_i} \right) - \frac{2}{3} \left(\bar{\rho}\kappa + \frac{1}{Re_t} \frac{\partial \widetilde{u}_l}{\partial x_l} \right) \delta_{ij} \right] \frac{\partial \widetilde{u}_i}{\partial x_j} \ , \tag{23}$$

$$p^* = \bar{p} + \frac{2}{3} \left[\left(\frac{1}{Re} + \frac{1}{Re_t} \right) \frac{\partial \widetilde{u}_l}{\partial x_l} + \bar{\rho}\kappa \right] \ , \tag{24}$$

with the model constants given by:

$$C_\mu = 0,09 \ , \ C_{\varepsilon 1} = 1,44 \ , \ C_{\varepsilon 2} = 1,92 \ , \ C_{\varepsilon 3} = 0,288 \ , \ \sigma_\kappa = 1 \ , \ \sigma_\varepsilon = 1,3 \ , \ Pr_t = 0,9 \ .$$

2.2 NEAR WALL TREATMENT

The $\kappa - \epsilon$ model is incapable of properly representing the fluid behavior in the laminar sub-layer and in the transition region of the turbulent boundary layer. To solve this inconvenience, the standard solution is the use of wall laws, capable of properly representing the flow in the inner region of the turbulent boundary layer. There are four velocity and two temperature laws of the wall implemented in the Turbo2D code, in wich one temperature and three velocity wall laws are sensible to pressure gradients. In this work, considering that no significative pressure gradients are involved, only the logarithm law is used. The logarithm law of the wall for velocity is already well known, and further explanations are unnecessary.

For the near wall temperature, Cheng and Ng (1982) derived an expression similar to logarithmic law of the wall for velocity. For the numerical calculation purposes, the intersection point between laminar and logarithmic sub-layers are defined at $y^* = 15,96$, with $y^* = u_f \delta / \nu$, where u_f is the friction velocity calculated by the relation

$$u_f = \left(\frac{1}{Re} + \frac{1}{Re_T} \right) \frac{\partial u_i}{\partial x_j} - \frac{1}{\rho} \frac{\partial P}{\partial x_i} \delta_{ij} \tag{25}$$

ν is the kinetic viscosity and δ is the distance until the wall. The temperature wall laws for laminar and logarithmic sub-layers are respectively

$$\frac{(T_0 - T)_y}{T_f} = y^* \, Pr \quad \text{and} \quad \frac{(T_0 - T)_y}{T_f} = \frac{1}{K_{Ng}} ln \ y^* + C_{Ng} \ , \tag{26}$$

where T_0 is the environmental temperature and T_f is the friction temperature, defined by the relation

$$T_f u_f = \left[\left(\frac{1}{Re \, Pr} + \frac{1}{Re_t \, Pr_t} \right) \frac{\partial \widetilde{T}}{\partial x_j} \right]_\delta . \tag{27}$$

In the equation (26) the constants K_{Ng} and C_{Ng} are, respectively, 0,8 and 12,5. The turbulent Prandtl number Pr_t is assumed constant and equal to 0,9.

For the turbulent kinetic energy κ and for the rate of dissipation of the turbulent kinetic energy ε, the near wall values are taken by the following relations

$$\kappa = \left[\frac{{u_f}^2}{\sqrt{C_\mu}} \right]_\delta \quad \text{and} \quad \varepsilon = \left[\frac{{u_f}^3}{K\delta} \right]_\delta , \tag{28}$$

with $K = 0,419$.

3. NUMERICAL METHODOLOGY

The calculation domain adopted in this work, for both simulations is described bellow.

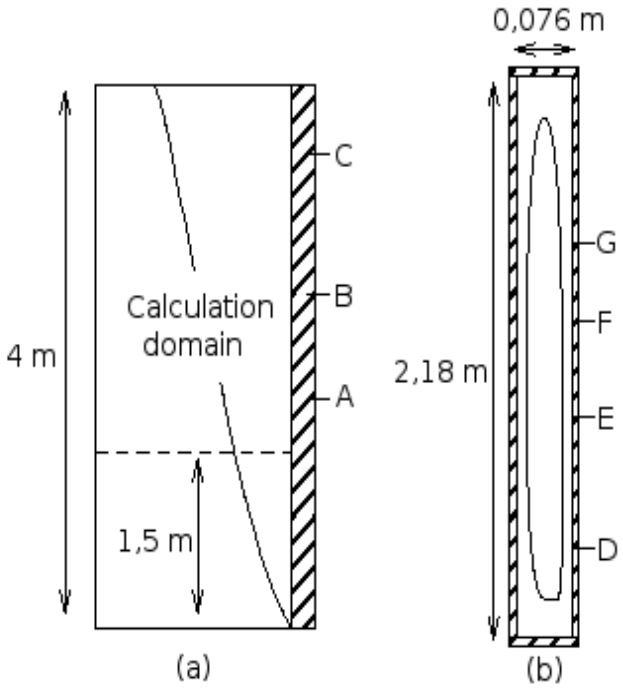

Figure 1. Calculation domain. (a) Nagano et. al (1992) test case and (b) Betts and Bhokari (1996) test case

The figure 1(a) represents the calculation domain for the Nagano et.al (1992) test case. The points A,B and C, represent the sections where the profiles were taken, their heights are respectively 1,92m, 2,54m and 3,24m. The boundary inlet conditions for this test case are: velocity, temperature, κ and ε experimental profiles. In the outlet atmospheric pressure condition was setted up. The right wall was heated up to a constant temperature of 60^{o}C. The boundary wall velocity was calculated with the classic logarithmic wall law, in a distance about $y^+ = 4.8$. For the left wall a null derivative condition for all the variables was adopted. This decision was based in the fact that the fluid is acelerated as it passes throught the heated wall, since the size of the domain is constant, by the continuity principle, some mass must be injected in the domain, and in this case, it is done by it's lateral in the left border of the domain. With the adoption of this boundary condition, the solver was capable to determinate this lateral flow, as it will be shown in figure 3.

The figure 1(b) represents the calculation domain for the Betts and Bhokari (1996) test case. The points D,E,F and G represent the sections where the profiles were taken, theis heights are respectively 0,66m, 1,09m, 1,31m, and 1,53m. It is important to note that this test case consists in a recirculating flow, so there isn't an inlet or an outlet. The boundary conditions in the lower and upper part of the domain are constant temperature gradients. The side walls are maintained at constant temperatures. For the right wall a temperature of 36 ^{o}C was imposed, while for the left wall a temperature of 15 ^{o}C was setted up. The boundary wall velocity was calculated with the classic logarithmic wall law, in a distance about $y^+ = 2$.

The meshes used in the simulation consists in a P1 mesh with 3360 nodes and 6460 elements and a isoP2 mesh with 13179 nodes and 25840 elements for the Nagano et. al (1992) test case, and a P1 mesh with 4199 nodes and 7920 elements with a isoP2 mesh with 16317 nodes and 31680 elements for the Betts and Bhokari (1996) test case.

It is important to say that a mesh study was done in order to define the necessary refinement level of the meshes used in this work. The isoP2 meshes for both test cases are shown in the figure 2.

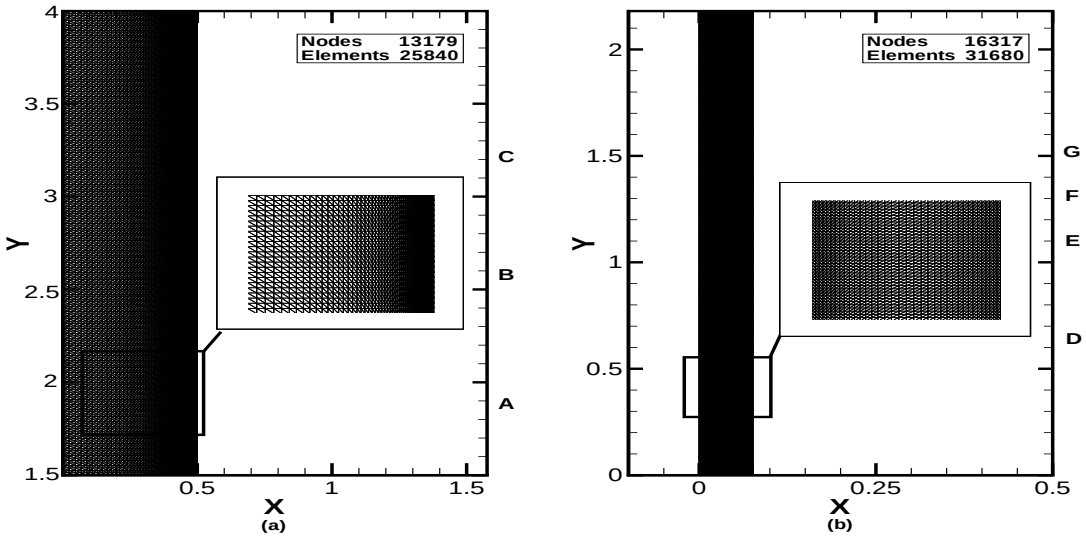

Figure 2. Meshes used in the simulations. (a) Nagano et. al (1992) test case and (b) Betts and Bhokari (1996) test case

4. NUMERICAL RESULTS

Some qualitative results for both test cases are shown in figure 3.

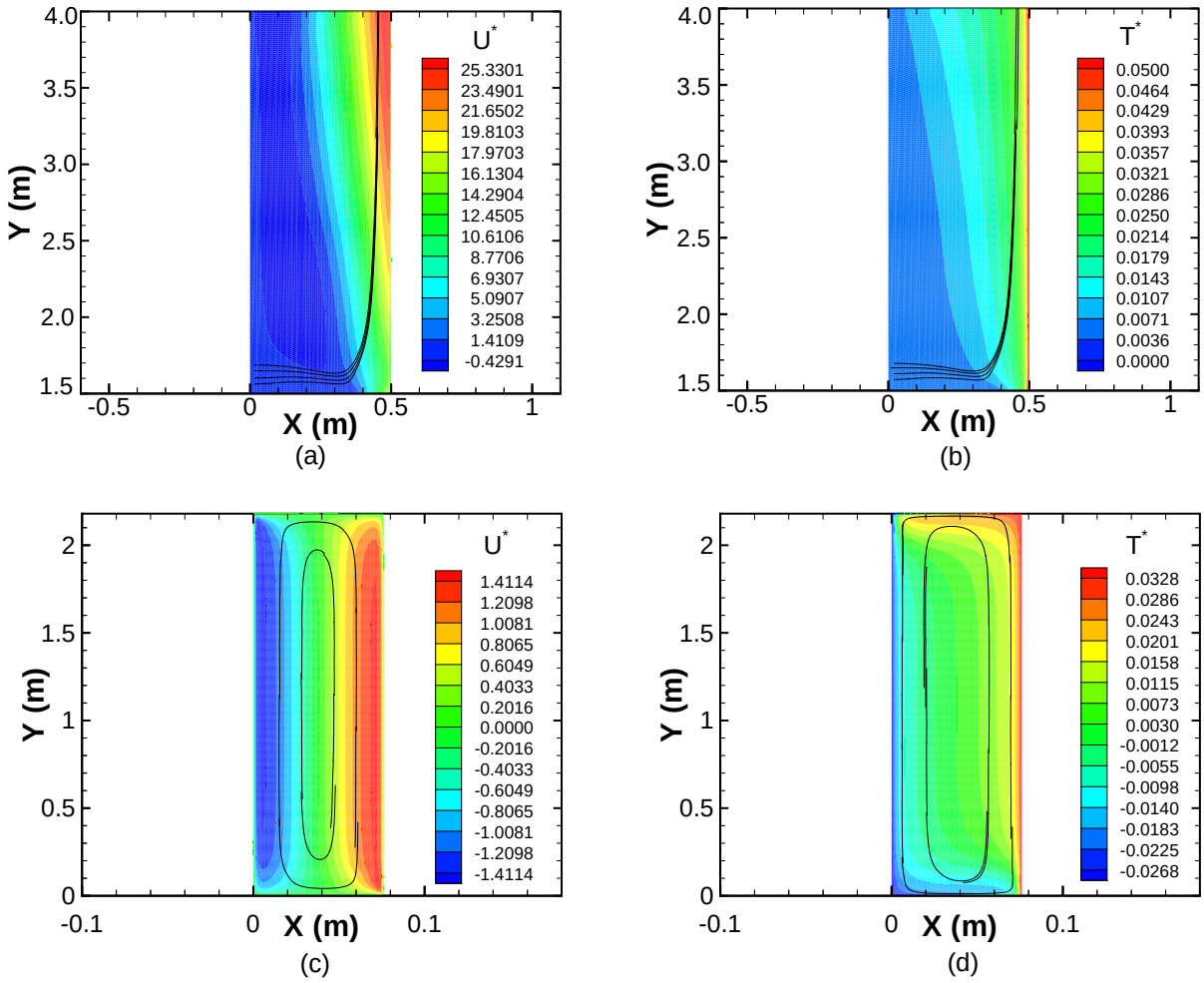

Figure 3. Nagano et. al (1992) test case - velocity field (a), temperature field (b). Betts and Bhokari (1996) test case - velocity field (c), temperature field (d)

It can be observed from the heated vertical flat plate studied by Nagano (1992), by the streamlines presented in figures 3a and 3b that a lateral flow exists. This flow is explained by the continuity principle, since the fluid is accelerated as it flows through the plate, and the calculation area domain is constant, some mass must be injected in the domain, and it's done by it's lateral, with the null derivative velocity boundary condition imposed in the left border of the domain, the solver is capable to calculate this lateral flow. In the test case of Betts and Bhokari, the great recirculating region can be observed, by the streamlines presented in figures 3c and 3d. The values in the legend are in the non dimentional form

$$U^* = \frac{U}{U_0} \qquad \text{and} \qquad T^* = \frac{T - T_0}{T_0} \tag{29}$$

where the index $_0$ indicates reference values and * represent the non dimention form of the component. In figure 4, the velocity and temperature profiles, taken for the test case of Nagano (1992) are shown, followed by the analysis of the Nusselt number along the plate in figure 5.

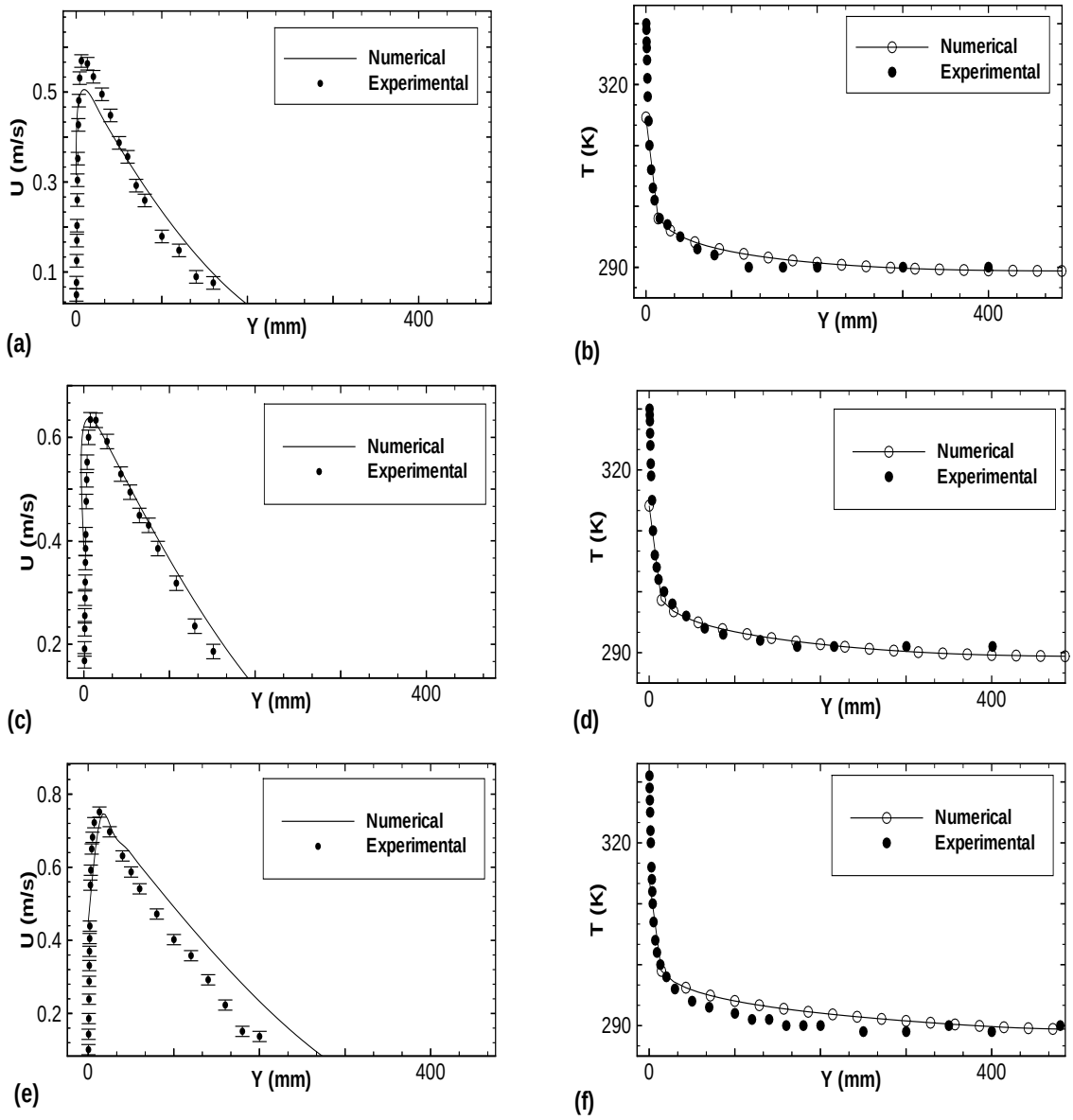

Figure 4. Nagano et.al (1992) test case - Velocity and temperature profiles, taken at the heights of 1,92m (a),(b), 2,54m (c),(d), and 3,24m (e),(f)

In a general way, the numerical modeling of the velocity and temperature field in the Nagano et. al (1992) test case is good. The maximum values of velocity are well modeled in figure 4c and 4e (points B and C). In the figure 4a (point A) the maximum velocity value is underestimated. In figure 4e (point C), despite the good prediction of the maximum velocity

value, a superestimated velocity profile is observed as the distance from the wall is raised. The numerical modeling obtained for the temperature field presents in figures 4b,4d,4e, a good agreement with the experimental values. In figure 5, the heat flux along the plate for the Nagano et.al (1992) test case is represented by the variation of the local Nusselt number as a function of the Rayleigh number, both defined as:

$$Nu_x = \frac{hx}{k} \qquad \text{and} \qquad Ra_x = \frac{g\beta(T_w - T_\infty)x^3}{\nu\alpha} \tag{30}$$

In equation (30), β is the thermal expantion coefficient, α is the thermal difusivity, and x is the local coordinate in the flow direction. It is possible to notice that the numerical values obtained for the local Nusselt number are under the experimental uncertainty interval along all the observed domain.

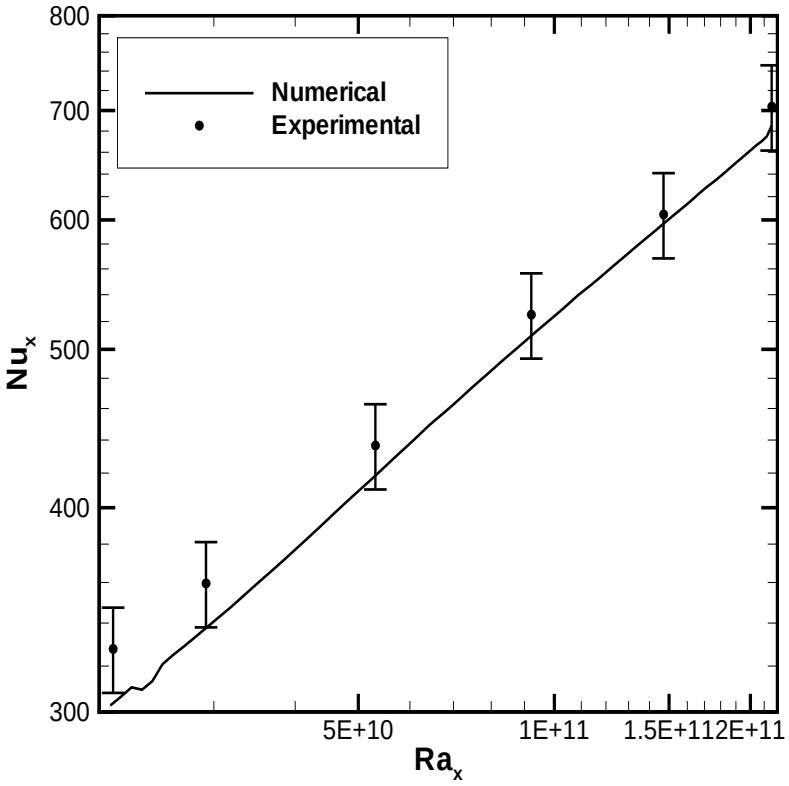

Figure 5. Nagano et. al (1992) test case - Local heat flux, quantified in a Nusselt x Raighley graphic

In the work of Betts and Bhokari (1996), the main idea was to measure the thermal and dynamic behavior of a recirculating flow inside a tall cavity, in wich turbulence in low Reynolds number occurs. Figures 6 and 7 show velocity and temperature profiles for the Betts and Bhokari (1996) test case.

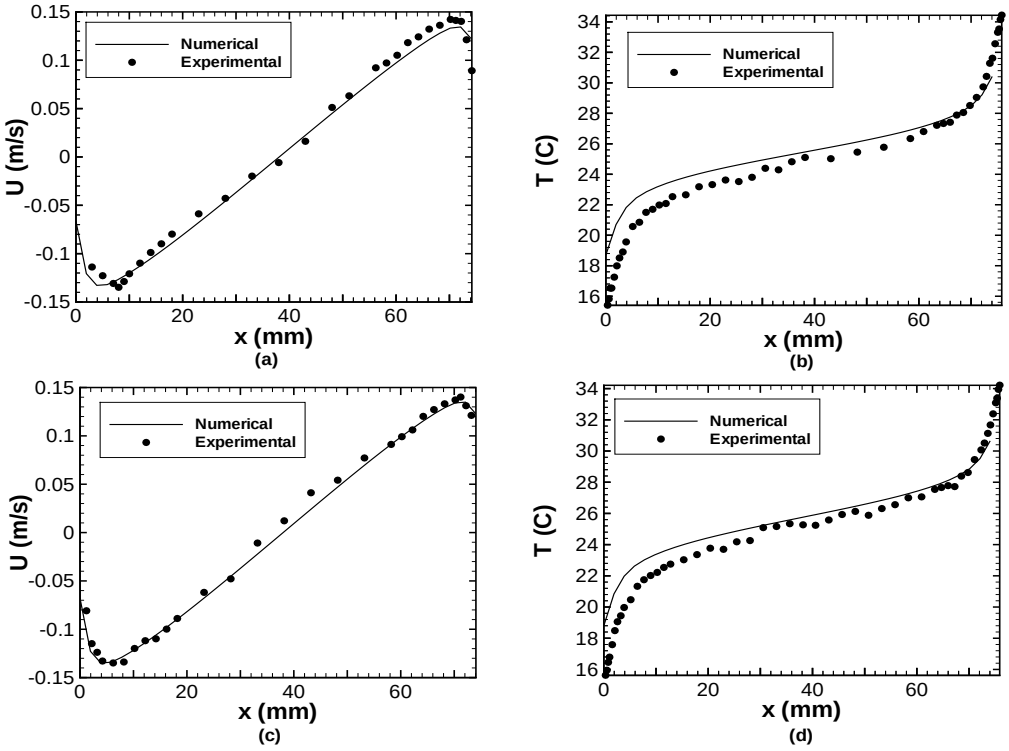

Figure 6. Betts and Bhokari (1996) test case - Velocity and temperature profiles, taken at the heights of 0,66m (a),(b) and 1,09m (c),(d)

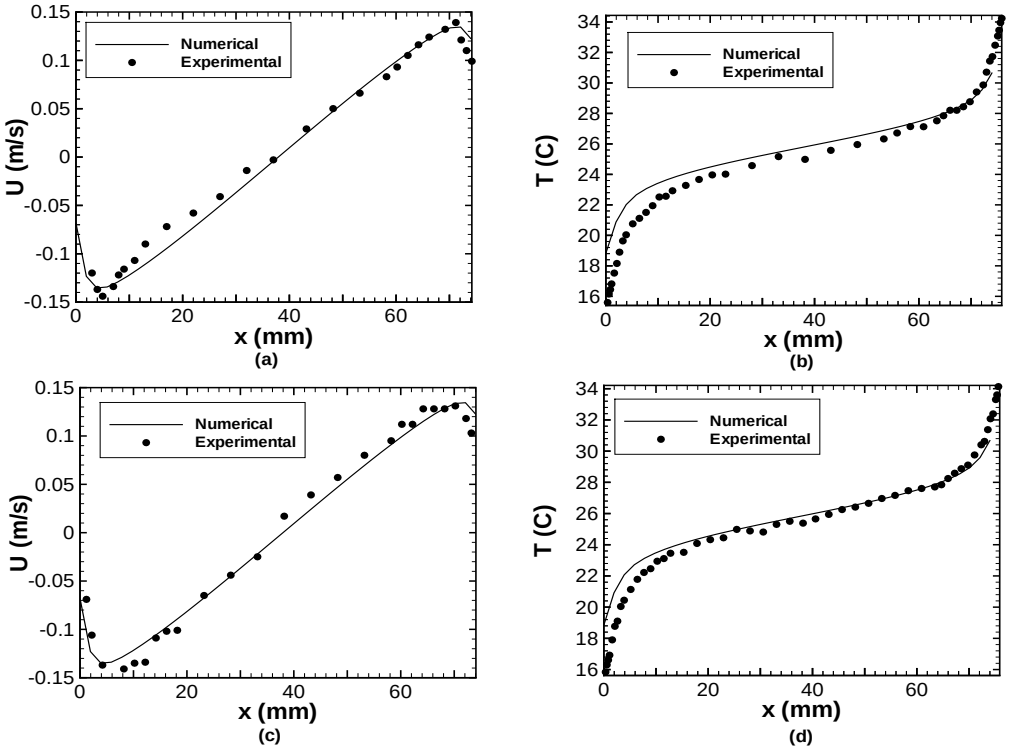

Figure 7. Betts and Bhokari (1996) test case - Velocity and temperature profiles, taken at the heights of 1,31m (a),(b) and 1,53m (c),(d)

The numerical results of the velocity and temperature profiles obtained for the Betts and Bhokari (1996) test case show a good agreement with the experimental data in all the analyzed sections. In this test case the velocity profiles are closer to the experimental values than the velocity profiles.

5. CONCLUSIONS

The velocity, temperature profiles and the wall heat flux for the Nagano et.al (1992) test case, obtained numericaly are in a good agreement with the experimental data in both test cases, as shown in the figures 4,5,6 and 7.

The lack of a better agreement between the experimental and the numerical velocity profile in figure 4e (point C) of Nagano et. al (1992) test case, may be atributed to the uncertainty in the boundary velocity condition in the left side of the calculation domain, due to existence of a lateral flow.

The determination of the local Nusselt number shows that the differences between the numerical and the experimental values are under the experimental uncertainty interval in all the observerd domain, showing that the numerical methodology employed can reproduce well the heat fluxes measured experimentaly.

The qualitative results shows that the velocity contours and the streamlines represents very well the great recirculation zone inside the cavity of Betts and Bhokari (1996), and the lateral mass flux in the vertical heated flat plate of Nagano et. al (1992).

In a general way the numerical methodology employed in the Turbo2D code produces a good numerical modeling of cases involving turbulent natural convection over flat geometries, such as in vertical walls (test case of Nagano et.al) as in recirculating cavities (test case of Betts and Bhokari).

6. REFERENCES

ABCM, 2004, "Journal of the Brazilian Society of Engineering and Mechanical Sciences", http://www.abcm.org.br/journal/index.shtml. Boussinesq, J., 1877, "Théorie de lt'Écoulement Tourbillant" , Mem. Présentés par Divers Savants Acad. Sci. Inst. Fr., vol. 23, pp. 46-50.

Betts and Bhokari, 1996, "Turbulent Natural Convection in an Enclosed Tall Cavity". Ercoftac, case 79.

Brun, G., 1988, "Developpement et application dt'une methode dt'elements finis pour le calcul des ecoulements turbulents fortement chauffes " , Doctorat thesis, Laboratoire de Mécanique des Fluides, Escola Central de Lyon.

Buffat, M., 1981, "Formulation moindre carrés adaptées au traitement des effets convectifs dans les équation de Navier-Stokes", Doctorat thesis, Université Claude Bernard, Lyon, France.

Brison ,J. F., Buffat, M., Jeandel,D., Serrer,E., 1985, "Finite elements simulation of turbulent flows, using a two equation model", Numerical methods in laminar and turbulent flows, Swansea. Pineridg Press.

Cheng, R.K. and Ng, T.T., 1982, "Some aspects of strongly heated turbulent boundary layer flow". Physics of Fluids, vol. 25(8).

Colburn, A.P., 1933 , "A method for correlating forced convection heat transfer data and a comparison with fluid friction " Transaction of American Institute of Chemical Engineers, vol. 29, pp. 174-210.

Cruz, D.O.A., Silva Freire, A.P., 1998, "On single limits and the asymptotic behavior of separating turbulent boundary layers", International Journal of Heat and Mass Transfer, vol. 41, n^o 14, pp. 2097-2111

Favre, A., 1965, "Equations de gaz turbulents compressibles". Journal de mecanique, vol. 3 e vol. 4.

Fontoura Rodrigues, J. L. A., 1990, "Méthode de minimisation adaptée à la techinique des éléments finis pour la simulation des écoulements turbulents avec conditions aux limites non linéaires de proche paroi", Doctorat thesis, Ecole Centrale de Lyon, France.

Jones, W. and Launder, B.E., 1972, "The prediction of laminarization with a two equations model of turbulence", International Journal of Heat and Mass Transfer, vol. 15, pp. 301-314.

Kays,W.M., Crawford, M.E. 1993, "Convective Heat and Mass Transfer", McGraw Hill, INC., USA

Launder, B.E. and Spalding, D.B., 1974, "The numerical computation of turbulent flows", Computational Methods in Applied Mechanical Engineering, vol. 3, pp. 269-289

Mellor, G.L., 1966, "The effects of pressure gradients on turbulent flow near a smooth wall", Journal of Fluid Mechanics, vol. 24, n^o 2, pp. 255-274

Nakayama, A., Koyama, H., 1984, "A wall law for turbulent boundary layers in adverse pressure gradients", AIAA Journal, vol. 22, n^o 10, pp. 1386-1389

Reynolds, O., 1895, "On The Dynamical Theory of Incompressible Viscous Fluids and the Determination of the Criterium" , Philosophical Transactions of the Royal Society of London, Series A, Vol 186, p. 123

Tsuji, T. and Nagano, Y., 1988. "Turbulence Measurements in a Natural Convection Boundary Layer along a Vertical Flat Plate.", Int. J. Heat Mass Transfer, Vol. 31, No.10, 2101-2111

7. Responsibility notice

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