Proceedings of ENCIT 2008
12th Brazilian Congress of Thermal Engineering and Sciences
November 10-14, 2008, Belo Horizonte, MG

IMPLEMENTATION AND VALIDATION OF A METHODOLOGY USED TO IMPOSE HEAT FLUX BOUNDARY CONDITION IN A HIGH REYNOLDS TURBULENCE MODEL

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

Abstract. *The main goal of this work is to develop, implement and validate a numerical methodology to be used in the imposition of heat fluxes boundary conditions in high Reynolds turbulence models, such as the classical κ - ε. In order to achieve this goal, a test case of a thermal backward facing step was used to calibrate an expression capable to adjust the behavior of the Stanton number inside the recirculating region. Informations given by the use of classical analogies combined with the physical reality of the backward facing step problem and the use of laws of the wall are employed for the development of the algorithm used to impose this kind of thermal condition in problems with and without boundary layer deattachment. The algorithm of numerical resolution used to execute the simulations applies a consolidate Reynolds and Favre averaging process for the turbulent variables, and uses the classical κ - ε model. The internal regions of the velocity and thermal boundary layer are 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 coupling pressure-velocity 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 explicit applications of velocity laws of wall, are treated by a minimal residual method.*

Keywords: *turbulence model, heat transfer, thermal backward facing-step, wall laws, analogies*

1. INTRODUCTION

The numerical simulation of turbulent flows using high Reynolds turbulence models, such as the classical $\kappa - \varepsilon$, is one of the most popular methods used to describe the main turbulent variable fields of a great range of industrial interest flows. The greatest advantages of this approach are the low computational cost of the simulations and a considerable good quality of the results obtained. One of the most remarkable aspects of this kind of methodology is the use of mathematical expressions to predict the fluid flow behavior in a region wich is very closed to the wall. These expressions are the laws of the wall and are used to reduce the computational cost, since the high gradients involved in this region would demand very refined meshes. Meanwhile the use of laws of the wall is also responsible for the lose of information in a certain area of the calculation domain, witch can be a great disadvantage depending on the interest of the numericist. In the simulation of turbulent flows with thermal field, a temperature law of the wall is required. When the boundary condition in the solid wall is expressed in terms of temperature there are no difficulties, since there are a considerable number of temperature laws of the wall, but when the boundary condition is expressed in terms of the local heat flux, a problem occurs, since a heat flux law of the wall is still not available.

The main goal of this work is to propose, implement and validate a numerical methodology to be used in the simulation of turbulent thermal flows with local heat flux boundary conditions. In order to achieve this goal, an algorithm is proposed, based on the use of classical analogies to quantify the numerical value of the local convective heat flux coefficient working simultaneously with temperature and velocity laws of the wall.

Mathematical analogies between the local friction coefficient and the Stanton number have been studied for a long time, that interest can be observed in the works of Reynolds (1874), Colburn (1933), Von Karman (1939), Schultz-Grunow (1941), Martinelli (1947), Reynolds et. al (1958) and Kays et. al (1993). The results obtained by the use of classical analogies, such as the Colburn (1933) analogy for turbulent flows without boundary layer deattachment, proved to be in a good range of agreement with experimental and numerical results, as showed by Gontijo and Fontoura Rodrigues (2006a and 2006b). By the other hand, in deattached boundary layers, the use of the same analogies produces very underestimated values of the local Stanton number in the inside of the recirculation region as described by Gontijo and Fontoura Rodrigues (2007b).

In this work an adjust function for the Stanton number, with validity in the recirculation region, was developed using the thermal backward facing-step problem studied by Vogel and Eaton (1985) as the physical reality to calibrate this function. The created algorithm was capable to produce great results for the temperature profiles obtained numerically in the case of deattached turbulent boundary layers, even in the recirculation region. The explanation of the main idea of the proposed algorithm will be better understand in the next sections.

The solver used to execute the simulations is called Turbo2D, witch is a research Fortran numerical code, that has been continuously developed 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 technic, under the formulation of pondered residuals proposed by Galerkin, adopting in the spatial discretization of the calculation domain 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 turbulent variables ceases. The temporal discretization 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 velocity and temperature profiles obtained numerically are compared to the experimental data of Vogel and Eaton (1985).

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,$\|g\|$ is the absolute value of the gravity acceleration vector, 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. The lenght used in the definition of Re and Fr in this case is the height of the backward facing step.

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:

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

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:

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

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

where the viscous stress tensor is

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

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

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

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 $-\overline{\rho}\overline{u_i''u_j''}$, and by the fluctuations of temperature and velocity, the so-called turbulent heat flux, defined by the vector $-\overline{\rho}\overline{u_i''T''}$.

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

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

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

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

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:

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

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

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

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

$$\frac{\partial\left(\bar{\rho}\varepsilon\right)}{\partial t}+\widetilde{u}_i\frac{\partial\left(\bar{\rho}\varepsilon\right)}{\partial x_j} = \frac{\partial}{\partial x_i}\left[\left(\frac{1}{Re}+\frac{1}{Re_t\,\sigma_\varepsilon}\right)\frac{\partial\varepsilon}{\partial x_i}\right] +\frac{\varepsilon}{\kappa}\left(C_{\varepsilon 1}\Pi-C_{\varepsilon 2}\bar{\rho}\varepsilon+C_{\varepsilon 3}\frac{\bar{\rho}\beta g_i}{Re_t\,Pr_t}\frac{\partial\widetilde{T}}{\partial x_i}\right) \ , \tag{20}$$

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

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 the 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

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

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 $K=0,419$.

2.3 THE STANTON NUMBER

In many engineering practices the representation of important parameters are made in a non dimentional form. The wall heat flux, for example, can be estimated in a non dimension basis by using the local Stanton number, that can be calculated by two distinct manners. The first one is a classical way to turn the local parietal heat flux q_x in a dimensionless form:

$$St_x = \frac{q_x}{\rho c_p u_\infty (T_w - T_\infty)} \qquad \text{where, for a flat plate} \qquad q_x = -k\left(\frac{\partial T}{\partial y}\right)_{y=0}. \tag{29}$$

In equation (29) an accurate calculation of the temperature gradient is a difficult task since the use of wall laws give raise to some lost of some information in the wall region.

Another way to compute the local Stanton number is based in the use of analogies. An special analogy derived from a diversification of the Reynolds analogy, made by Colburn (1933) for fluids with the Prandtl number equal or larger than $0,5$, is called the Colburn analogy. The Colburn (1933) empirical correlation establish a relationship between the local Stanton number St_x, the local friction coefficient C_{fx} and the Prandtl number Pr:

$$St_x = \frac{C_{fx}}{2Pr^{\frac{2}{3}}}. \tag{30}$$

In equation (30), the local friction coefficient Cf_x is calculated with the use of the friction velocity u_f, numerically generated by the Turbo2D code:

$$\frac{C_{fx}}{2} = \frac{\tau_w}{\rho u_\infty^2} \qquad \text{with} \qquad \tau_w = \rho u_f^2 \qquad \text{so} \qquad \frac{C_{fx}}{2} = \frac{u_f^2}{u_\infty^2}. \tag{31}$$

2.4 PROPOSED ALGORITHM

The algorithm here proposed, to be used in turbulent thermal flows with heat flux specified as the wall boundary condition, is based in a numerical methodology the can be resumed by the following steps:

1. The value of the local friction velocity u_f calculated by eq.(25) is used to determine the value of the local friction coefficient.

   $$Cf_x = \frac{2u_f^2}{u_\infty^2} \tag{32}$$

   In eq.(32), u_∞ represents the velocity outside the boundary layer.

2. The value of Cf_x is then used to calculate the local Stanton number using the Colburn(1933) analogie as expressed in eq.(33).

   $$St_x = \frac{Cf_x}{2Pr^{2/3}} \tag{33}$$

3. By the definition of the Stanton number it is possible to estimate the value of the local convective heat flux coefficient, with eq.(34).

   $$h_x = St_x \rho C_p u_\infty \tag{34}$$

   Where ρ is the fluid density and C_p represents the specific heat at a constant pressure.

4. With the value of h it is possible to do an aproximatelly conversion of the local heat flux into the local wall temperature. This information is then sended to the temperature law of the wall that calculates the temperature boundary condition in the first node of the mesh.

The main idea of this algorithm is to use the values of dynamical parameters of the flow in the wall, such as the friction velocity to estimate the heat transfer rates, with the use of classical analogies, in order to convert an imposed heat flux in the wall into a calculated wall temperature, so the thermal law of the wall can calculates the temperature boundary condition in the wall nodes.

It is important to say that for recirculation regions, a correction need to be done in the value of the local Stanton number. In this work it was created an adjust function based on the data of an experiment of Vogel and Eaton (1985) in a thermal backward facing-step with a thermal boundary condition on the wall of a constant heat flux imposed. This special treatment will be better explained in the future sections.

3. NUMERICAL METHODOLOGY

The numerical solution of a dilatable turbulent flow, has as main difficulties: the coupling between the pressure, velocity and temperature fields; the non-linear behavior of the momentum and energy equations; the explicit calculations of boundary conditions in the solid boundary; the methodology of use the continuity equation as a manner to link the coupling fields of velocity and pressure.

The solution proposed in the present work suggests a temporal discretization of the system of governing equations with a sequential semi-implicit finite difference algorithm proposed by Brun (1988) and a spatial discretization using finite elements of the type P1-isoP2. The temporal and spatial discretization implemented in Turbo 2D is presented in Fontoura Rodrigues (1990).

4. NUMERICAL MODELING

The test case used to achieve the proposed goal is based in the thermal backward facing step of Vogel and Eaton (1985). The schematic representation of the calculation domain is described in figure (1)

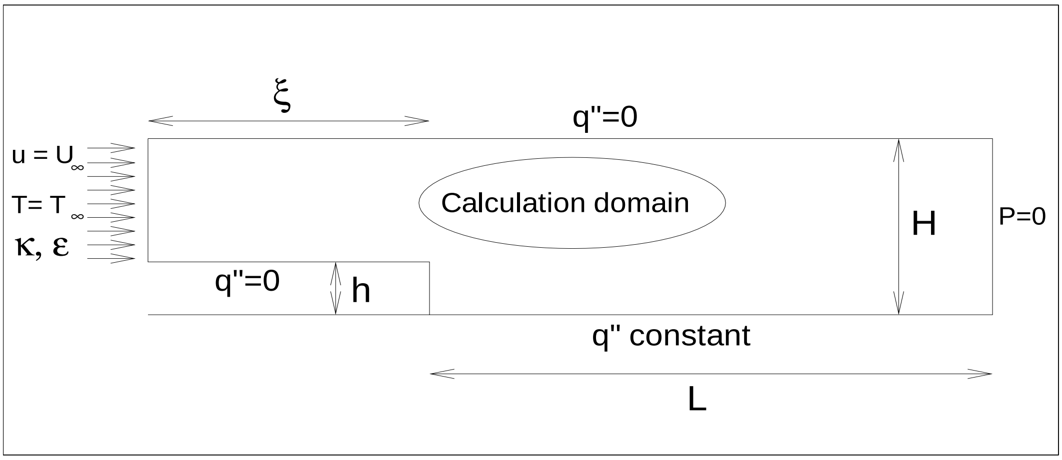

Figure 1: calculation domain

Figure (2) displays the meshes used to execute the simulation.

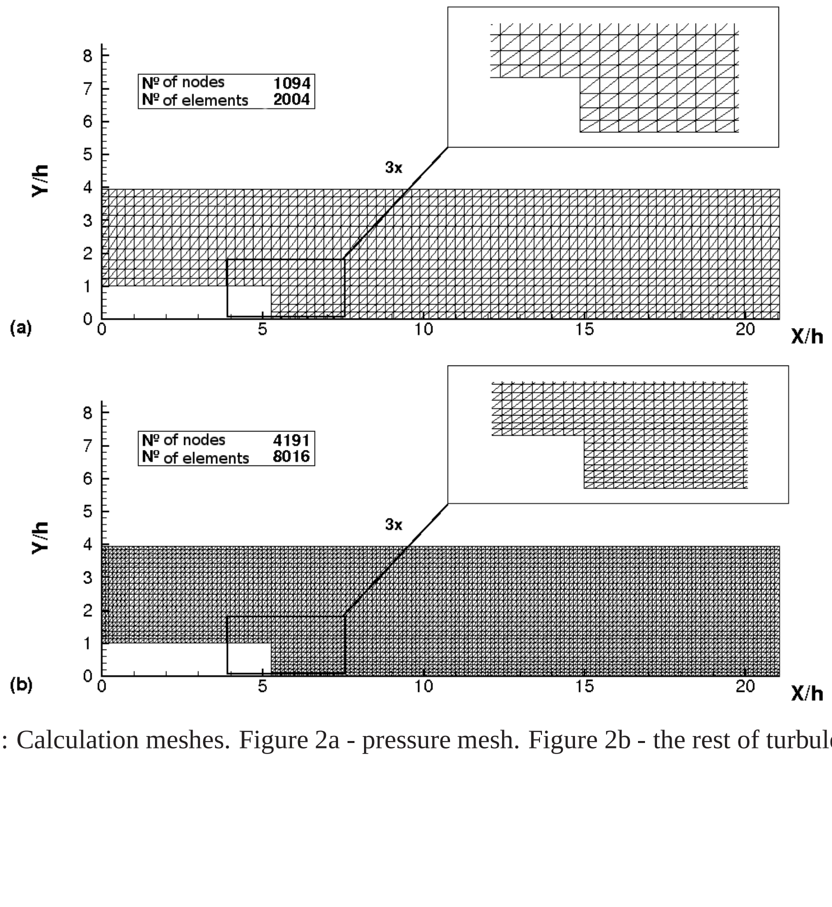

Figure 2: Calculation meshes. Figure 2a - pressure mesh. Figure 2b - the rest of turbulent variables

The dimentions in figure (1) are: h = 0,038 m, H = 0,15 m, ξ = 0,2 m and L = 0,6 m. In the inlet where imposed flat velocity and temperature profiles with the purpose of allow the flow development before the deattachment point, since there is some uncertainty about the experimental inlet profiles. κ and ε values in the inlet where estimated based on the turbulence level of the wind tunnel described by Vogel and Eaton (1985). In the lower wall of the backward facing step a condition of constant heat flux was imposed and in the top wall and adiabatic condition was estabilished. In the outlet the flow is in atmospheric pressure. The Reynolds number based on the height of the step is 27023. The velocity on the free stream flow is 11,3 m/s and the heat flux imposed is 270 W/m^2. The low value of the imposed heat flux was setted to avoid significant variations of the thermodynamic properties of the fluid by the increase of temperature, such as the fluid density, even though in the numerical resolution, a dilatable formulation was used in order to produce a more accurate value of the dynamical and thermal field.

Figure (2.a) shows the P1 mesh used to calculate the presure field, while figure (2.b) ilustrates the P1/isoP2 mesh, that was used to calculate all the other variables, such as velocity, temperature, fluid density, κ and ε. It is possible to notice that 8016 finite elements were used to calculate the main turbulent variables. Indeed, 8 thousand elements, for this domain, constitute a reasonable refined mesh. The final dimension of the mesh was obtained with a mesh study. This number of elements were able to provide a numerical simulation with a low computational cost and a good quality of the results, as the next sections will show. It is also important to notice a greater refinement level in the near wall elements. This procedure is important, since higher gradients appear in this region.

The pressure and kinetic energy fields obtained numerically are showed in figure (3) and (4).

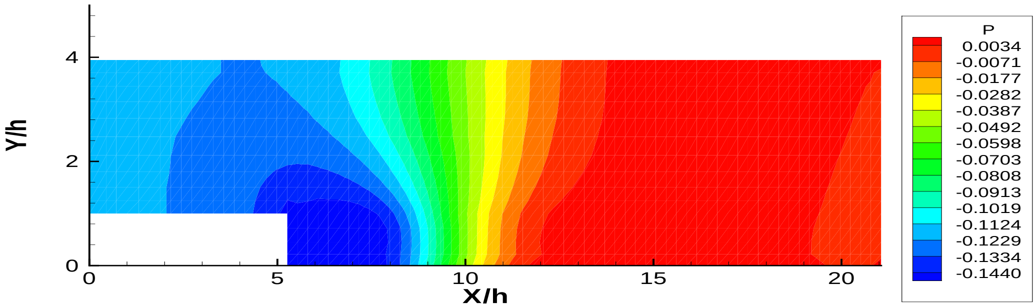

Figure 3: pressure field

The pressure field in figure (3) shows a low pressure zone in the recirculation region and a change in the signal of pressure in the reattachment point, this behavior is classic in deattached boundary layers. The values in the legend represent the non dimentional pressure, defined in equation (5).

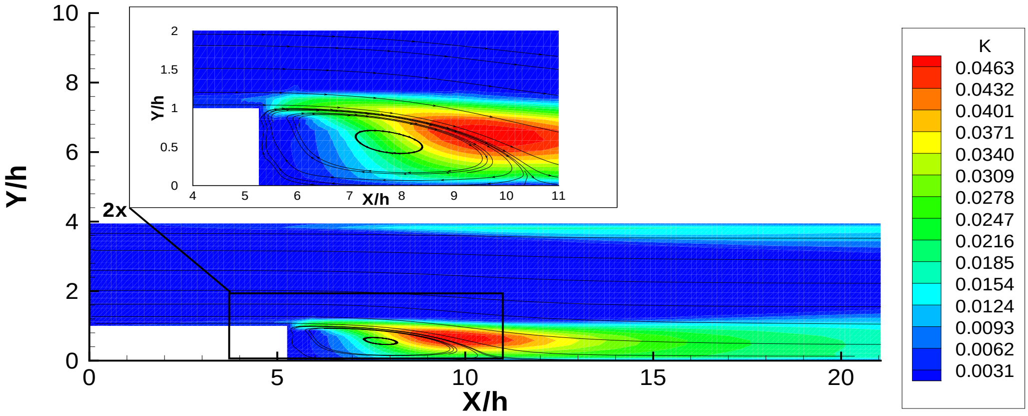

Figure 4: kinetic energy field

In figure (4) is possible to observe low values of κ in the deattachment point, those values increase until provide an intense kinetic energy field near the reattachment point. This behavior is classic in backward facing steps simulations. The black lines represents the streamlines of the flows. The experimental reattachment point occurs in $x/h = 10,4$ and the obtained numerical value is $x/h = 10,6$. These informations are significant to conlude that the dynamical parameters of the flow are in a good agreement with experimental data.

The Stanton number behavior through the lower wall was numerically calculated with the use of the Colburn (1933) analogy, equation(33), by using the values of the friction velocity u_f to estimate the Stanton number. Figure (5) shows that the numerical values inside the recirculation region have the same qualitative behavior of the experimental values, but are different by a scale factor. The numerical Stanton number in figure (5a) was calculated with equation (33).

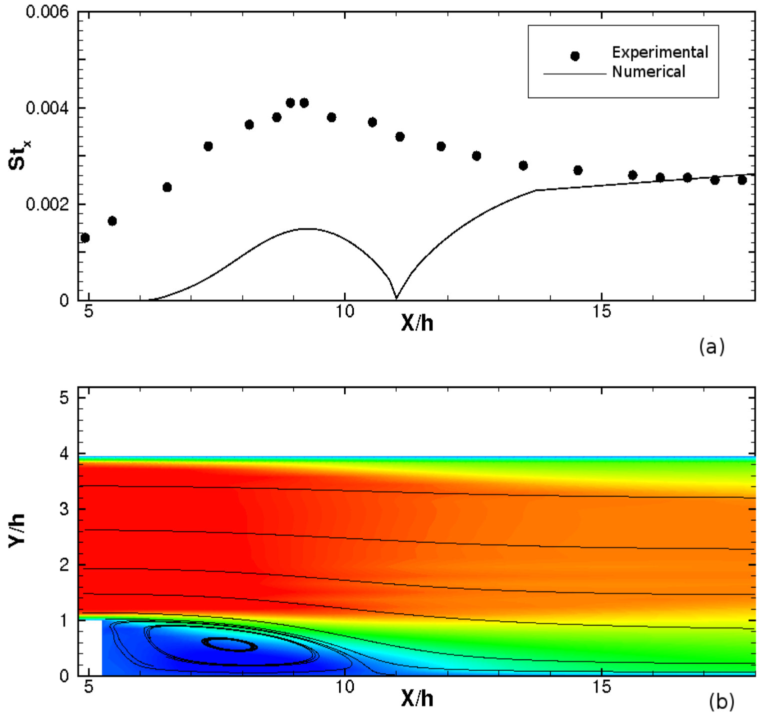

Figure 5: Numerical and experimental behavior of the Stanton number (a), velocity field and streamlines of the flow (b)

It is possible to notice that outside the recirculation zone, the numerical and experimental values are very close, and that the higher values of the Stanton number occur near the reattachment point. This behavior is directly related to the kinetic energy field ilustrated in figure (4), since turbulence converts great part of the kinetic energy of the flow into heat. The numerical values outside the recirculation region are very closed to the experimental behavior.

It is also possible to notice that in the reattachment point the numerical value of St_x goes to values near zero, since the shear stress in the wall in this point is very small and consequently the values of Cf_x. This shows that the use of classical analogies are not appropriated in flows with boundary layer deattachment, as showed by Gontijo and Fontoura Rodrigues (2007b). Even recognizing that the use of classical analogies inside deattached boundary layers have serious limitations, the behavior showed in figure (5a) suggests the adoption of an adjust factor to the Stanton number inside this region. It is important to notice that the observed values of St_x will change with the distance between the wall and the first node of the mesh. By this reason, it was used the average value of y^+, that express the distance to the wall in witch the simulation is done. The definition of y^+ is

$$y^+ = \frac{\overline{u_f}\delta}{\nu} \tag{35}$$

where $\overline{u_f}$ is the average value of the friction velocity calculated through the wall of the recirculation region, δ is the distance from the wall and the first node of the mesh and ν is the kinematic viscosity of the fluid.

With the definition of y^+, several simulations were executed with different values of y^+, and an adjust constant was calibrated for each simulation, with this methodology an adjust function was created and it's behavior is well shaped by a power law as expressed in equation (36)

$$f(y^+) = 5,46y^{+^{-0,2936}}. \tag{36}$$

With the aid of the correction shape function, the proposed equation for the calculation of the Stanton number inside recirculation regions is the modifyed Colburn analogie

$$St_x = max\left(\frac{Cf_x}{2Pr^{2/3}}5,46y^{+^{-0,2936}};0,02\right) \tag{37}$$

Figure (6) shows how the modifyed Colburn analogie works. In the recirculation region, the discontinuity in the Stanton number in the reattachment point is avoided and the magnitude order of the values is very similar to the experimental data. This methodology allows the simulation of turbulent thermal flows with heat flux boundary condition using a high Reynolds turbulence model. Some points can still be improved in future works, and this methodology needs more study in order to extend this treatment to other complex geometries.

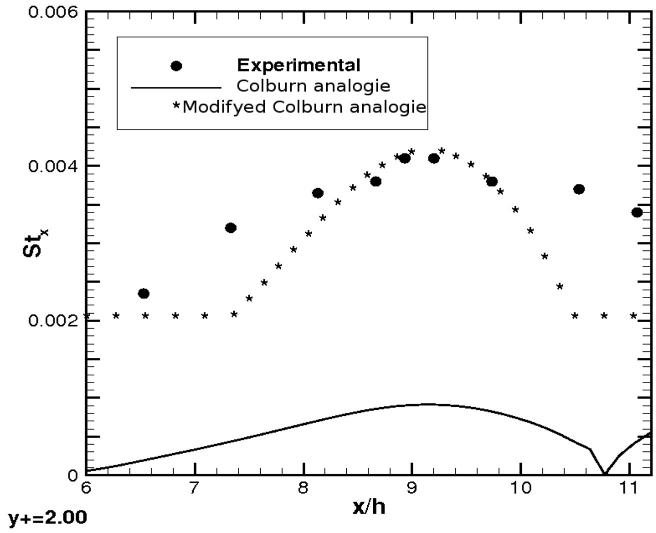

Figure 6: adjust of the Stanton number inside the recirculation region

The most important validation of this methodology are the temperature profiles obtained for this test case, that are displayed in figure (7).

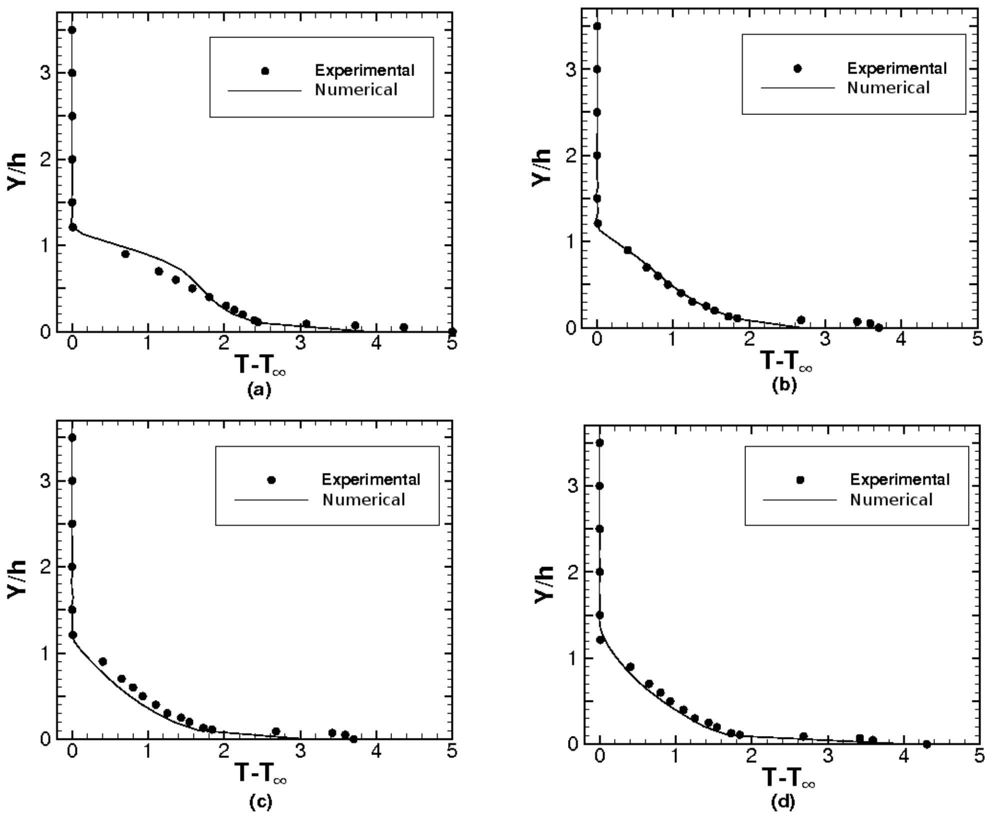

Figure 7: temperature profiles at positions: x/h=8.73 (a),x/h=10.87 (b),x/h=13.00 (c) and x/h=17.23 (d)

In order to give an idea of the sections where the profiles were taken, figure (8) shows the calculation domains with the streamlines and the geometric places where numerical and experimental profiles were compared.

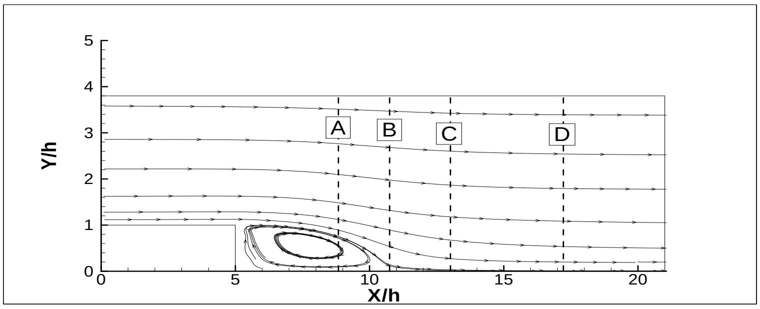

Figure 8: Sections where the profiles were taken

Figure (7) shows a good aproximation between the numerical and experimental values of the temperature profiles in the lower wall of the thermal backward facing step. Theses profiles are considered a good validation of the proposed methodology specially in the interior of the recirculation region, as displayed by figures (7a) and (7b).

5. CONCLUSIONS

The proposed algorithm was capable to reproduce with a good precision the temperature field of a turbulent flow of air over a thermal backward facing step with a heat flux boundary condition, using the classical $\kappa - \varepsilon$ model, as shown by figure (7).

The creation of an adjust function for the Stanton number inside the recirculation region, was based in an empyrical numerical philosophy witch is valid only for this situation. This work does not bring universality to the simulation of flows with heat flux boundary conditions using the $\kappa - \varepsilon$ model, but it is capable to give better results under certain conditions. Extrapolations of this methodology for other kinds of flow with boundary layer deattachment still needs mindful validation. Even though the use of the proposed algorithm should produce good results for flows without boundary layer deattachment, since Gontijo and Fontoura Rodrigues (2006a and 2006b) proved that the use of the Colburn analogy produces good results in turbulent flows over flat plates using the numerical value of the local friction velocity to estimate the local Stanton number.

It is important to say that near the deattachment and reattachment points, the law of the wall lose the capacity to calculate the wall boundary condition, since the shear stress in the wall goes to zero and so the friction velocity. In order to fix this situation, a numerical algorithm of error minimization based on the value of the friction velocity was developed by Fontoura Rodrigues (1990), and it makes possible the simulation of deattached boundary layer flows using the presented methodology.

Figure (6) shows a difference between the values of the local Stanton number obtained with the adjust function and the experimental ones. This difference can be reduced with the adoption of an alternative mathematical method of aproximation such as a spline fit for example, but is important to notice that even with this limitation the results obtained are good, as figure (7) ilustrates.

The results obtained in this work represent the begining of a greater study, wich the main goal is to extend this methodology to deattached boundary layer flows with distinct characteristics, especially for deattached boundary layers generated exclusive by the action of adverse gradient pressures, such as it occurs in conical diffusers and other geometries.

6. REFERENCES

Boussinesq, J., 1877, "Théorie de lt'Écoulement Tourbillant" , Mem. Présentés par Divers Savants Acad. Sci. Inst. Fr., vol. 23, pp. 46-50.

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.

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.

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.

Gontijo,R.G., Fontoura Rodrigues, J.L.A. 2006, "Numerical modeling of the heat transfer in the turbulent boundary layer", Cilamce and Encit, 2006

Gontijo,R.G., Fontoura Rodrigues, J.L.A. 2007, "Numerical modeling of a turbulent flow over a 2D channel with a rib-roughned wall", Cobem, 2007

Huges TJR, Brooks A; "A multi-dimentional upwind scheme with no crosswind diffusion", in *Finite Element Methods fo Convection Dominated Flows*, ASME - AMD 34. New York (1979)

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.

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

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

Kelly DW, Nakazawa S, Zienkiewiczs OC, Heinrich J; *"A note on upwind and anisotropic balancing dissipation in finite element aproximations to convective diffusion problems"*, International Journal for Numerical Method in Enginering, 15,11, pp.1705-1711 (1976).

Martinelli RC; "Heat transfer to molten metals," *Transactions of ASME* 69, pp.947-959 (1947)

Reynolds O; "On the extent and action of the heating surfaces for steam boilers," *Procedures of Manchester Literature Philosophical Society* 14,pp.7-12 (1874)

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

Reynolds, W.C., Kays, W.M. and Kline, S.J., 1958, "Heat Transfer in The Turbulent Incompressible Boundary Layer - II - Step Wall-Temperature Distribution", NASA Memorandum.

Schultz-Grunown F; "New frictional resistence law for smooth plates," NACA TM 986 (1941)

Vogel, J. C., Eaton, J. K., "Combined heat transfer and fluid dynamic measurements downstream of a backward-facing step", Journal of heat transfer; November 1985, vol. 107, pp 922-929

von Kármán T; "The analogy between fluid friction and heat transfer," *ASME Transactions* 61, pp.705-710 (1939)

7. Responsibility notice

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