THERMAL BOUNDARY CONDITIONS BASED ON THE USE OF ANALOGIES. A NUMERICAL STUDY.

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 use of classical analogies between turbulent heat and momentum diffusion, employed to estimate the wall heat transfer rates in turbulent wall flows, is numerically modelated in this work. Some approaches are proposed in order to quantify the heat transfer rates in recirculation regions of turbulent flows and an algorithm, used to impose heat flux boundary conditions in turbulent flows modeled with the classic* $\kappa - \varepsilon$ *model, is developed and validated. The proposed algorithm uses simultaneously a thermal law of the wall and an interpolating polynomial relation that is constructed with a data base generated on experimental research and in present numerical simulation. 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* $\kappa - \varepsilon$ *model. The turbulent inner layer can be modeled by four distinct velocity laws of the wall and by one thermal law of the wall. Spacial discretization is done by P1 and P1/isoP2 finite elements and the temporal discretization is implemented using a semi-implicit sequential scheme of finite differences. 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 explicit calbulations of laws of the wall, are treated by a minimal residual method.*

Keywords: *turbulence, laws of the wall, thermal boundary conditions, analogies, boundary layer detachment*

1. INTRODUCTION

The study of turbulent flows over solid surfaces is a domain of great industrial interest. In the last decades a huge amount of work has been done on the development of several turbulence models in order to better represent the dynamic behavior of complex turbulent flows. These new approaches are usually associated with a higher computational cost and even though the processing capacity of modern machines is increasing in an expressive speed, the requirements of industry are still for faster simulations with a good quality of results. By these reasons, according to Chen and Jaw (1998), the classic high Reynolds $\kappa - \varepsilon$ model is still the most used turbulence model in industry.

In terms of a relation between cost and benefit the $\kappa - \varepsilon$ model is capable to present good quality of results without being an expensive model. Among the reasons for the low cost of this model, is expressive the contribution of laws of the wall, capable to represent the fluid flow behavior in the internal region of the turbulent boundary layer. The adoption of laws of the wall allows the use of more coarse meshes, since the flow in the wall region, witch is dominated by the highest gradients and by the intricate mechanism of production and dissipation of turbulence is not simulated. The greatest disadvantage of this approach is the loss of information in the near wall region witch creates, in some cases, additional complications in the imposition of certain boundary conditions.

In flows heated by the presence of a wall on witch is imposed a specified heat flux, the imposition of this boundary condition is particular complicated with the high Reynolds $\kappa - \varepsilon$ model, since the absence of a heat flux law of the wall compel us to use another approach, in order to simulate this kind of flows.

A possible way to impose heat flux boundary condition, in a high Reynolds turbulence model, is to convert an imposed heat flux into an equivalent temperature on the wall, by estimating the convection heat transfer coefficient. The heat transfer rates in turbulent boundary layer flows can be estimated with a good precision by the use of analogies between momentum and heat diffusion, according to Gontijo and Fontoura Rodrigues (2006). In cases where the boundary layer is not well established, for example when occurs the boundary layer detachment, the use of analogies is not indicated and produces bad results as showed by Gontijo and Fontoura Rodrigues (2007).

A numerical methodology developed to impose heat flux boundary conditions in the high Reynolds $\kappa - \varepsilon$ model, in flows with and without boundary layer detachment, was proposed by Gontijo and Fontoura Rodrigues (2008). In the present work, an evolution of this methodology is developed, implemented and validated in three different test cases: the backward facing step of Vogel and Eaton (1985); the asymmetric plane diffuser of Buice and Eaton (1995); the hill of Loureiro et. al (2007).

The solver used, named Turbo2D, 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 technique, under the formulation of weighted residuals proposed by Galerkin, adopting in the spatial discretization of the calculation domain with the triangular elements of the type P1 and P1-isoP2, as proposed by Brison, Buffat, Jeandel and Serres (1985). The P1-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 P1-isoP2 mesh.

Considering the uncertainties normally existing about the initial conditions of the problems that are numerically simulated, it is adopted the temporal integration of the governing equations system. In the temporal integration 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, in order to reach the final state, a pseudo transient occurs. 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 Uzawa'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 tensor 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 four velocity and two temperature laws of the wall. The velocity laws of the wall used in this work are: the classical logarithm law, and the laws of Mellor (1966), Nakayama and Koyama (1984), and Cruz and Silva Freire (1998). The temperature law of the wall used is the Cheng and Ng (1982) law of the wall. The numerical instability resultant of the explicit calculation of the boundary conditions of velocity, through the evolutive temporal process, is controlled by the algorithm proposed by Fontoura Rodrigues (1990). The numerical oscillations induced by the Galerkin formulation, resultant 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 Vogel and Eaton (1985) and two qualitative test cases were proposed based on the works of Buice and Eaton (1995) and Loureiro et. al (2007).

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:

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

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:

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

$$\frac{\partial}{\partial t}\left(\overline{\rho}\widetilde{u}_i\right)+\frac{\partial}{\partial x_j}\left(\overline{\rho}\widetilde{u}_j\widetilde{u}_i\right)=-\frac{\partial\overline{p}}{\partial x_i}+\frac{\partial}{\partial x_j}\left[\overline{\tau_{ij}}-\overline{\rho u_j^{\prime\prime}u_i^{\prime\prime}}\right]+\overline{\rho}g_i, \tag{8}$$

where

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

$$\frac{\partial(\overline{\rho}\widetilde{T})}{\partial t}+\frac{\partial(\widetilde{u}_i\widetilde{T})}{\partial x_i}=\frac{\partial}{\partial x_i}\left(\alpha\frac{\partial\widetilde{T}}{\partial x_i}-\overline{\rho}\overline{u_i^{\prime\prime}T^{\prime\prime}}\right) \tag{10}$$

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

In these equations α is the molecular thermal diffusivity and two news unknown quantities appear in the momentum (8) and in the energy equations (10), defined by the correlations between the velocity fluctuations, the so-called Reynolds Stress, given by the tensor $-\overline{\rho}\overline{u_i^{\prime\prime}u_j^{\prime\prime}}$, and by the fluctuations of temperature and velocity, the so-called turbulent heat flux, defined by the vector $-\overline{\rho}\overline{u_i^{\prime\prime}T^{\prime\prime}}$.

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), wheresuggest

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

with

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

and

$$\mu_t=C_\mu\overline{\rho}\frac{\kappa^2}{\varepsilon}=\frac{1}{Re_t} \quad . \tag{14}$$

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

$$-\overline{\rho}\overline{u_i^{\prime\prime}T^{\prime\prime}}=\frac{\mu_t}{Pr_t}\frac{\partial\widetilde{T}}{\partial x_i}. \tag{15}$$

In the equation (14) C_μ is a constant of calibration of the model, that values $0,09$, κ represents the turbulent kinetic energy and ε is the rate of dissipation of the turbulent kinetic energy. 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:

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

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

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

$$\begin{aligned}
\frac{\partial\left(\bar{\rho}\kappa\right)}{\partial t}+\widetilde{u}_i\frac{\partial\left(\bar{\rho}\kappa\right)}{\partial x_j} &= \frac{\partial}{\partial x_i}\left[\left(\frac{1}{Re}+\frac{1}{Re_t\,\sigma_\kappa}\right)\frac{\partial\kappa}{\partial x_i}\right]+\Pi-\bar{\rho}\varepsilon+\frac{\bar{\rho}\beta g_i}{Re_t\,Pr_t}\frac{\partial\widetilde{T}}{\partial x_i}\ , \qquad (19)\\
\frac{\partial\left(\bar{\rho}\varepsilon\right)}{\partial t}+\widetilde{u}_i\frac{\partial\left(\bar{\rho}\varepsilon\right)}{\partial x_j} &= \frac{\partial}{\partial x_i}\left[\left(\frac{1}{Re}+\frac{1}{Re_t\,\sigma_\varepsilon}\right)\frac{\partial\varepsilon}{\partial x_i}\right]\\
&\quad +\frac{\varepsilon}{\kappa}\left(C_{\varepsilon 1}\Pi-C_{\varepsilon 2}\bar{\rho}\varepsilon+C_{\varepsilon 3}\frac{\bar{\rho}\beta g_i}{Re_t\,Pr_t}\frac{\partial\widetilde{T}}{\partial x_i}\right)\ , \qquad (20)\\
\bar{\rho}\left(1+\widetilde{T}\right) &= 1\ , \qquad (21)
\end{aligned}$$

where:

$$\begin{aligned}
\frac{1}{Re_t} &= C_\mu\bar{\rho}\frac{\kappa^2}{\varepsilon}\ , \qquad (22)\\
\Pi &= \left[\left(\frac{1}{Re_t}\right)\left(\frac{\partial\widetilde{u}_i}{\partial x_j}+\frac{\partial\widetilde{u}_j}{\partial x_i}\right)-\frac{2}{3}\left(\bar{\rho}\kappa+\frac{1}{Re_t}\frac{\partial\widetilde{u}_l}{\partial x_l}\right)\delta_{ij}\right]\frac{\partial\widetilde{u}_i}{\partial x_j}\ , \qquad (23)\\
p^* &= \bar{p}+\frac{2}{3}\left[\left(\frac{1}{Re}+\frac{1}{Re_t}\right)\frac{\partial\widetilde{u}_l}{\partial x_l}+\bar{\rho}\kappa\right]\ , \qquad (24)
\end{aligned}$$

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 $\kappa-\varepsilon$ turbulence model is incapable of properly representing the laminar sub-layer and the transition regions of the turbulent boundary layer. To solve this inconvenience, the solution adopted in this work is the use of laws of the wall for temperature and for velocity, capable of reasonably representing the flow in the inner region of the turbulent boundary layer.

There are four velocity and two temperature laws of the wall implemented on Turbo 2D. In this work was used just the classic logarithmic law for velocity and the the law of the wall of Cheng and Ng (1982) for temperature, witch is explained bellow.

2.2.1 Temperature law of the wall of Cheng and Ng (1982)

For the calculation of the temperature, Cheng and Ng (1982) derived an expression for the near wall temperature similar to the logarithmic law of the wall for velocity. For the laminar and turbulent regions, the equations are respectively

$$\frac{\left(T_0-T\right)_y}{T_f}=y^*\,Pr \quad \text{and} \quad \frac{\left(T_0-T\right)_y}{T_f}=\frac{1}{K_{Ng}}ln(y^*)+C_{Ng} \qquad \text{with} \quad y^*=\frac{u_f y}{\nu} \qquad (25)$$

where T_0 is the environmental temperature, y is the normal distance up to the wall, ν is the cinematic viscosity and T_f is the friction temperature, as defined by Brun (1988)

$$T_f=\frac{1}{u_f}\left[\left(\frac{1}{RePr}+\frac{1}{Re_TPr_T}\right)\frac{\partial\widetilde{T}}{\partial x_j}\right]_\delta , \qquad (26)$$

and the friction velocity u_f is calculated by the relation

$$u_f=\left(\frac{1}{Re}+\frac{1}{Re_T}\right)\frac{\partial u_i}{\partial x_j}-\frac{1}{\rho}\frac{\partial P}{\partial x_i}\delta_{ij}. \qquad (27)$$

The intersection of these regions are at $y^*=15,96$ and the constants K_{Ng} and C_{Ng} are, respectively, $0,8$ and $12,5$.

2.3 THE STANTON NUMBER

In many engineering practices the representation of important parameters are made in a non dimensional 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 the definition of the Stanton number, wich is a representation in a dimensionless form of the local wall heat flux q_x:

$$St_x=\frac{q_x}{\rho c_p u_\infty(T_w-T_\infty)} \qquad \text{where, for a flat plate} \qquad q_x=-k\left(\frac{\partial T}{\partial y}\right)_{y=0}. \qquad (28)$$

In equation (28) an accurate calculation of the temperature gradient is a difficult task since the use of laws of the wall produce the loss 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

$$St_x = \frac{C_{fx}}{2Pr^{\frac{2}{3}}}. \tag{29}$$

In equation (29), the local friction coefficient Cf_x is calculated with the use of the friction velocity u_f, equation (27), numerically calculated by the Turbo2D code as

$$\frac{C_{fx}}{2} = \frac{\tau_w}{\rho u_\infty^2} \qquad \text{with} \qquad \tau_w = \rho u_f^2 \qquad \text{so} \qquad C_{fx} = 2\frac{u_f^2}{u_\infty^2}. \tag{30}$$

2.4 PROPOSED ALGORITHM

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

1. The value of the local friction velocity u_f calculated by equation (27) is used to determine the value of the local friction coefficient.

   $$Cf_x = 2\frac{u_f^2}{u_\infty^2} \tag{31}$$

   In equation (31), u_∞ represents the velocity outside the boundary layer or a caracteristic scale of velocity.

2. The value of Cf_x is then used to calculate the local Stanton number, using the Colburn(1933) analogie as expressed in eq.(32).

   $$St_x = \frac{Cf_x}{2Pr^{2/3}} \tag{32}$$

3. By the definition of the Stanton number it is possible to estimate the value of the local convective heat flux coefficient, with eq.(33).

   $$h_x = St_x \rho C_p u_\infty, \tag{33}$$

   where C_p represents the specific heat at a constant pressure.

4. With the value of h_x it is possible to do an approximately conversion of the local heat flux into an equivalent wall temperature. This information is then sent to the temperature law of the wall that calculates the temperature boundary condition in each wall 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 must be done in the value of the local Stanton number, since the detachment of the boundary layer cause to perish of analogies between heat and momentum diffusion. In the following sections is shown the evolution of the original methodology proposed by Gontijo and Fontoura Rodrigues (2008), with the numerical corrections that must be done in the local Stanton number.

3. NUMERICAL MODELING AND RESULTS

The test case used to develop the proposed methodology is the thermal backward facing step of Vogel and Eaton (1985). The schematic representation of the calculation domain is described in figure (1). The dimensions in figure (1) are: h = 0,038 m, H = 0,15 m, ξ = 0,2 m and L = 0,6 m. At the inlet of the calculation domain where imposed flat velocity and temperature profiles with the purpose of allow the flow development before the detachment point, since there is some uncertainty about the experimental inlet profiles. The κ and ε values, at the inlet, where estimated based on the turbulence level of the wind tunnel described by Vogel and Eaton (1985). At the wall of the backward facing step a condition of constant heat flux is imposed and in the top wall, an adiabatic condition was established. At the outlet of the calculation domain the gage pressure of the flow is null. The Reynolds number based on the height of the step is 27023. The free stream velocity 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 of the flow.

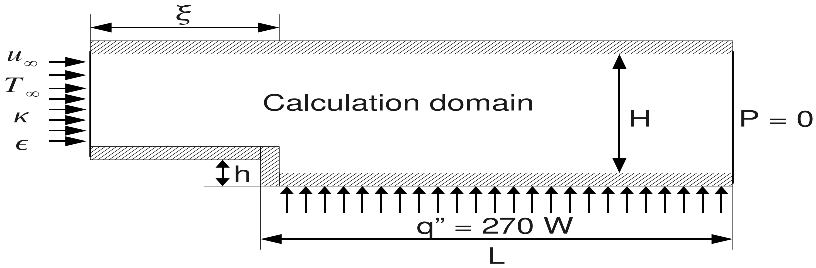

Figure 1. Calculation domain - Vogel and Eaton (1985) test case

The P1-isoP2 mesh used to execute the simulation has 8016 elements, while the P1 mesh contains 2004 elements. Indeed, eight 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 say that a greater refinement level was adopted in the near wall elements. This procedure is important, since higher gradients appear in this region.

The Stanton number behavior through the lower wall was numerically calculated with the use of the Colburn (1933) analogy, equation(32), by using the values of the friction velocity u_f to estimate the Stanton number. Figure (2a) 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.

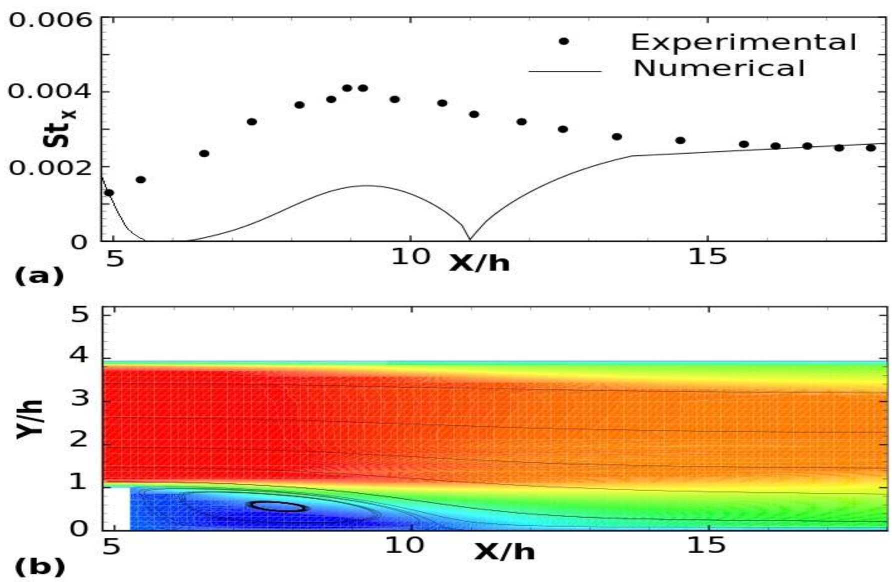

Figure 2. Numerical and experimental behavior of the Stanton number (a), velocity field and streamlines of the flow (b)

The analysis of the numerical simulation of Vogel and Eaton test case make evident the following features: outside the recirculation zone the numerical and experimental values are very close; the higher values of the Stanton number occur near the retachment point; the numerical values of St_x and Cf_x goes to values near zero at the retachment point, since the shear stress in the wall is very small at this locus. This shows that the use of classical analogies are not appropriated in flows with boundary layer detachment, as showed by Gontijo and Fontoura Rodrigues (2007b).

Even recognizing that the use of classical analogies inside detached boundary layers have serious limitations, the qualitative behavior shown in figure (2a) suggests that the adoption of an adjust factor to the Stanton number, inside this region, may improve the quality of those results. 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, the function to adjust the Stanton number in the detached boundary layer region, must have like independent variable the distance between the wall and the first node of the mesh. Our choice was to use a non dimensional form of the maximum value of the distance to the wall, y^+_{max}, defined by the relation

$$y^+_{max} = \frac{\overline{u_f}_{max}\delta}{\nu}, \tag{34}$$

where $\overline{u_f}_{max}$ is the highest value of the friction velocity calculated through the wall nodes of the calculation domain, δ is the distance between the wall and the first node of the mesh and ν is the kinematic viscosity of the fluid.

With the definition of y^+_{max}, several simulations were executed with different values of y^+_{max}, 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 by the following equation:

$$f(y^+) = 5,46{y^+_{max}}^{-0,2936}. \tag{35}$$

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

$$St_x = max\left(\frac{Cf_x}{2Pr^{2/3}}5,46{y^+_{max}}^{-0,2936}; 0,0128\right). \tag{36}$$

With this approach the discontinuity in the Stanton number, at the reattachment point, is avoided and the magnitude order of the numerical 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. The most important validation of this methodology are the temperature profiles obtained for this test case, this results were published by Gontijo and Fontoura Rodrigues (2008). It was obtained a good approximation between the numerical and experimental values of the temperature profiles in the lower wall of the thermal backward facing step. This means that this initial approach, at least, enables the use of heat flux boundary conditions in detached flows simulated with the high Reynolds $\kappa - \varepsilon$ model.

The first approach to adjust the Stanton number inside the recirculation region can still be improved, as can be seen in the figure (3), by this reason a second approach was developed. Therefore a non dimensional variable x^* was defined by the relation

$$x^* = \frac{x - x_d}{x_r - x_d}, \tag{37}$$

where x defines the local coordinate in the flow direction, x_d is the detachment point and x_r is the attachment point. After defining the variable x^*, an interpolation of the experimental values of the Stanton number measured by Vogel and Eaton (1985) inside the recirculation region was done and it produced the following relation

$$St\left(x^*\right) = 0,00128 + 0,00458x^* + 0,00321{x^*}^2 - 0,00574{x^*}^3. \tag{38}$$

After implementing equation (38) in the numerical code, the obtained behavior of the Stanton number is illustrated in figure (3).

Figure (3) shows that the second approach is capable to produce good values of the local Stanton number inside the recirculation region. It also shows that after the retachment point a sudden change in the numerical value of the local Stanton number occurs and that the numerical values recover the experimental behavior after a region with approximately the same length as the recirculation region. Basically what happens is that after the retachment point, the Colburn (1933) analogy is used directly to calculate the Stanton number. Analogies between heat and momentum diffusion are developed for well established boundary layers. After the retachment of the flow, the boundary layer is being restructured and in this region is expected that the Colburn (1933) analogy doesn't work well.

By this reason, a third approach was proposed. This last approach proposes that the interpolation defined in the second approach, must be extrapolated until the point after the retachment where the boundary layer is well established. In this

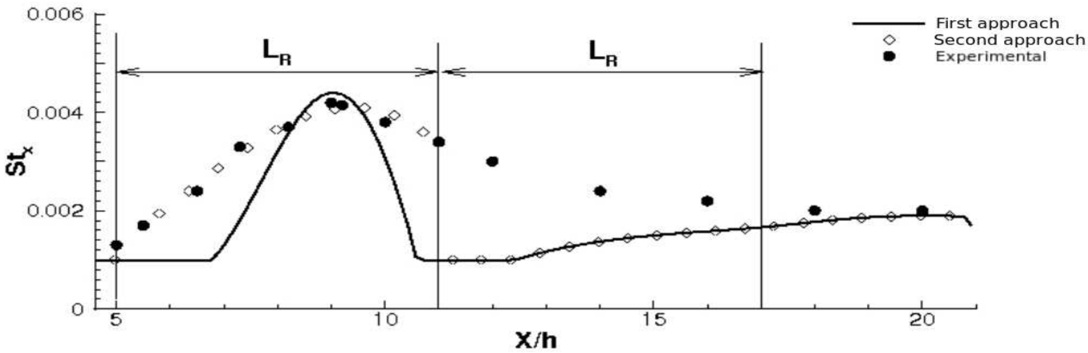

Figure 3. Adjusts obtained by the first and second approach

work, in the absence of best criterion, was arbitrarily considered that the necessary length for this proccess to occur is the own recirculation region length. The third approach calculates the local Stanton number by the following relation

$$St\,(x^*) = 0,00106 + 0,00912x^* - 0,00895x^{*2} + 0,00233x^{*3}. \tag{39}$$

The adjust obtained with this approach is illustrated in figure (4).

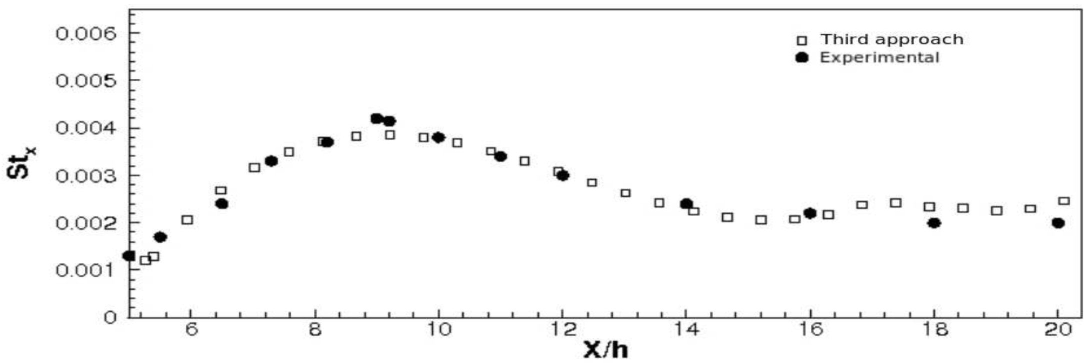

Figure 4. Adjusts obtained by the third approach

This is the best approach obtained so far and it is capable to produce great results in this specific test case. In order to extend this methodology to other geometries, two new test cases were proposed: the asymmetric plane diffuser of Buice and Eaton (1995) and the turbulent flow over a 2D hill, studied by Loureiro et. al (2007). The boundary conditions used to execute these simulations are illustrated in figure (5).

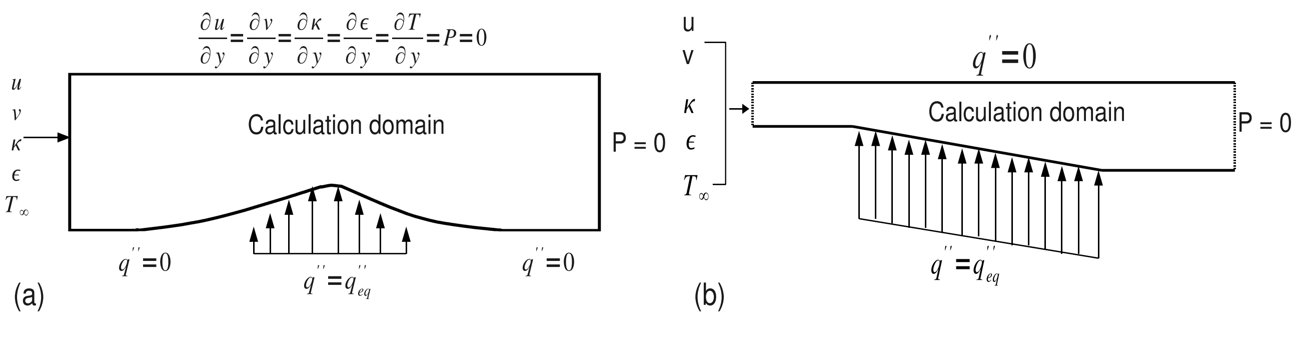

Figure 5. Geometry and boundary conditions of the Buice and Eaton (1995) diffuser (a), geometry and boundary conditions of the Loureiro et. al (2007) 2D hill (b)

In the experimental works of Buice and Eaton (1995) and Loureiro et. al (2007) the thermal field is not considered. They studied only the dynamical field. What was done to create two new test cases based on these experimental works was to calculate first the dynamical field of these flows, without inputing any thermal boundary condition. Then, a simulation with an imposed constant temperature on the wall was executed. After this step, the equivalent thermal energy injected in the flow is calculated by measuring the temperature profiles before and after the heated wall. Then an equivalent heat flux was calculated and imposed in the same wall, where the constant temperature condition was imposed. By doing this procedure is expected that the same energy injected in the flow, by the constant temperature boundary condition, should be injected by the equivalent constant heat flux condition, if the numerical method used to impose this condition is accurate. It is important to say that in both cases occur the boundary layer detachment. In order to obtain an accurate behavior of the velocity and temperature fields inside the recirculation regions, the law of the wall of Cruz and Silva Freire (1998) was used in both cases. This law was the one with the best performance among all the laws of the wall tested. These results were published in the master's dissertation of Gontijo (2009).

Figure (6) illustrates two temperature profiles taken after the heated wall for the Buice and Eaton (1995) test case and two for the 2D hill of Loureiro et. al (2007).

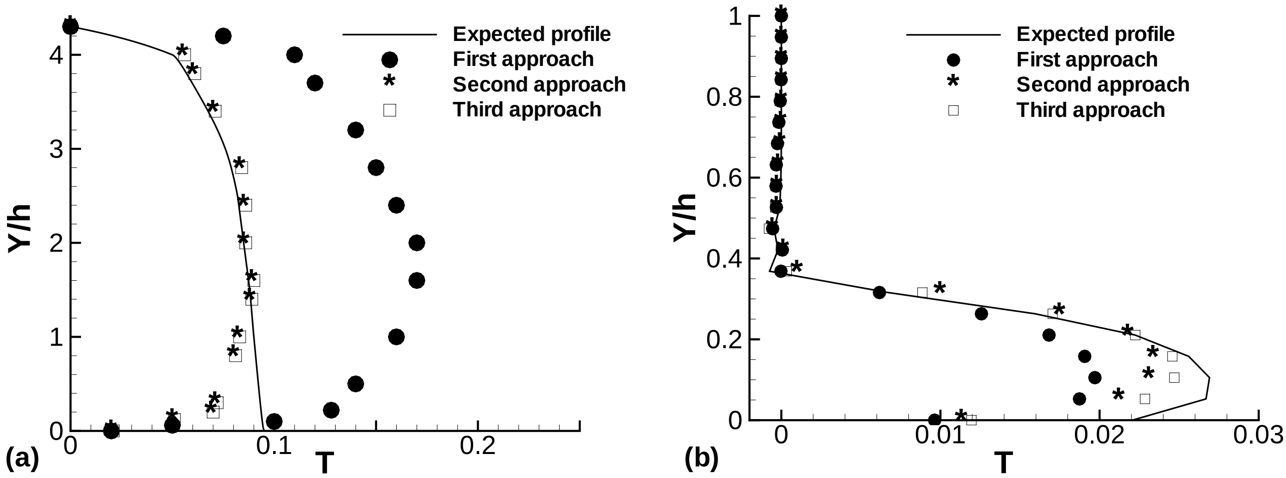

Figure 6. Temperature profiles in the asymmetric diffuser of Buice and Eaton (1995) - X/h=26 (a) and in the 2D hill of Loureiro et. al (2007) in X/h=6.5 (b)

It is possible to notice that the third approach is the final solution for the problem. In order to quantify the improvement obtained by the evolution of the initial approach, the average value of temperature was taken after the heated wall in four different points and compared for each approach with the obtained values for the constant temperature condition, this procedure generated table (1).

	Constant temperature	First approach	Second approach	Third approach
Buice and Eaton (1995) test case	326.05 K	342.03 K	320.53 K	320.87 K
Error	-	6,37%	0,32 % K	**0,21 %**
Loureiro et. al (2007) test case	300.07 K	299.54 K	299.84 K	299.89 K
Error	-	0.18%	0.08 % K	**0.06 %**

Table 1. Error comparison for Buice and Eaton (1995) and Loureiro et. al (2007) test cases

The results obtained with the final approach are good. This approach is able to predict the equivalent wall temperature inside the recirculation regions of flows over distinct boundary geometries, even when the mechanism responsible by the boundary layer detachment is a very smooth adverse pressure gradient.

4. CONCLUSIONS

This work proposed, implemented and validated a new and original numerical methodology used to impose heat flux boundary conditions in the high Reynolds $\kappa - \varepsilon$ model, without the need to create a heat flux law of the wall. Past works done by the authors were used to develop this methodology based on the employment of classical analogies between fluid friction and heat transfer on the wall. The test case used to develop this methodology in detached turbulent boundary layers and also to understand the main obstacles of this approach was the Vogel and Eaton (1985) backward facing step. The advances done based in this test case were then tested in two other different geometries and showed that this methodology can be extended to distinct geometries, even when the detached is induce by smooth adverse pressure gradients. One of the aspects that can be better studied is the necessary length to the restructuring of the boundary layer after the detachment, even though in the studied test cases the adopted criterium considered in this work has provided good results.

5. REFERENCES

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

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.

Buice, C. and Eaton, J., 1995. "Experimental investigationof flow through an asymmet- ric plane diffuser". Annual Research Briefs - 1995, Center of Turbulence Research, Stanford University/ NASA Ames. 117-120.

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.

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

Chen C.J. e Jaw, S.Y., 1998 "Fundamentals of Turbulence Modeling", Taylor and Francis, New York.

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.

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

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

Gontijo, R. G. e Fontoura Rodrigues, J.L.A., 2008, "Implementation and validation of a methodololgy used to impose heat flux boundary conditions in a high Reynolds turbulence model", Encit 2008.

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

Loureiro, J .B. R, Soares, D.V, Fontoura Rodrigues, J.L.A, Silva Freire, A.P, Pinho, F.T, "Water tank and numerical model studies of flow over 2-D hill", Boundary Layer Meteorology, Vol.122, Sï£¡rie 2 , pp.343-365, 2007.

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

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

6. Responsibility notice

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