Numerical Simulation of Turbulence in Open-Channel Flows

Daniel Vieira Soares
Departamento de Engenharia Mecânica - Universidade de Brasília - UnB
Campus Univeritário Darcy Ribeiro, CEP 70910-900, Brasília, Brasil
danielvs@linkexpress.com.br

Juliana Braga Rodrigues Loureiro
Programa de Engenharia Mecânica (PEM/COPPE/UFRJ)
C.P. 68503, 21945-970, Rio de Janeiro, Brasil
jbrloureiro@mecanica.coppe.ufrj.br

Átila Pantaleão da Silva Freire
Programa de Engenharia Mecânica (PEM/COPPE/UFRJ)
C.P. 68503, 21945-970, Rio de Janeiro, Brasil
atila@mecanica.coppe.ufrj.br

José Luiz Alves da Fontoura Rodrigues
Departamento de Engenharia Mecânica - Universidade de Brasília - UnB
Campus Univeritário Darcy Ribeiro, CEP 70910-900, Brasília, Brasil
fontoura@unb.br

Abstract. *The main goal of this work is to provide a performance analysis of a numerical simulation methodology in solving open-channel turbulent flows with separation of the turbulent boundary layer. To accomplish this, a selected experimental test case was chosen, a turbulent open-channel flow over a hill, where the detachment of the turbulent boundary layer occurs due to the presence of low intensity adverse pressure gradients. For example, this case is specially important to the research of turbulent mechanisms present in atmospheric flows over irregular terrains or obstacles, or river flows past submerged rocks. The mean equations of conservation of mass and momentum are obtained via the classic* $\kappa - \varepsilon$ *model of Jones and Launder. 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 of Galerkin's method to the convective fluxes, a balance dissipation method is adopted. The remaining non-linearities, due to the use of velocity laws of wall, are treated by a minimum residue method. The numerical results obtained with the use of four different velocity laws of the wall are compared to the available experimental data of the considered flow.*

Keywords: *turbulence, separating boundary layer, open-channel flow, laws of the wall, finite-elements method*

1. Introduction

The objective of this work is to test the numerical performance of different laws of the wall in cases of open-channel turbulent flows with separation of the boundary layer, testing the methodology shown by Soares and Fontoura Rodrigues (2003 and 2004) of numerical simulation using laws of the wall. To accomplish this, a selected experimental test case was chosen, a turbulent open-channel flow over a hill, where the detachment of the turbulent boundary layer occurs due to the presence of low intensity adverse pressure gradients, intercalated between two regions of stream-wise pressure gradients.

This case is specially important to the research of turbulent mechanisms present in atmospheric flows over irregular terrains or obstacles, or river flows past submerged rocks, all of which have several engineering applications, from seacraft design to atmospheric and climate forecasts.

The algorithm to be tested, Turbo 2D, is a combination of the numerical simulation methodology using finite elements, proposed by Brison et al. (1985), with an error minimization method adapted to a finite elements, for the simulation of turbulent wall flows with non-linear boundary conditions, proposed by Fontoura Rodrigues (1990 and 1991), using the classic $\kappa - \varepsilon$ turbulence model of Jones and Launder (1972). By applying Galerkin's method for finite elements to the calculation of convection dominant flows, numerical oscillations without physical meaning can appear. This fact occurs due to the usage of Galerkin's method, that gives a symmetric treatment to the flow modeling, which is a non symmetric physical phenomenon. To lower the tendency of numerical oscillation, a balancing dissipation method, proposed by Huges and Brooks (1979) and Kelly et al. (1980) and implemented by Brun (1988), is used in Turbo 2D.

The results obtained with simulations using four different laws of the wall, implemented in Turbo 2D, are compared to the experimental data available from the test case of Loureiro (2004): a turbulent open-channel flow over an abrupt hill.

2. Governing equations

The turbulent one phase flow analyzed in the present work is homogeneous, at low Mach number, and buoyancy forces are small compared to convective effects. Considering the procedures shown by Soares and Fontoura Rodrigues (2004), with pertinent adaptations for isothermal flows, the conservation equations of mass and momentum, which describe the phenomena, are respectively represented, in Einstein's notation, by the dimensionless relations:

$$\frac{\partial u_i}{\partial x_i} = 0 \ , \tag{1}$$

$$\frac{\partial u_i}{\partial t} + u_j \frac{\partial u_i}{\partial x_j} = -\frac{1}{\rho}\frac{\partial p}{\partial x_i} + \frac{1}{Re}\frac{\partial}{\partial x_j}\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right) \ , \tag{2}$$

where ρ is the fluid density, t represents time, x_i a cartesian coordinate, u_i is the i^{th} velocity component, p is the thermodynamic pressure and Re is the Reynolds number.

2.1 The turbulence model

The adopted methodology is a transformation of the system of instantaneous dimensionless governing equations into a system of mean equations, obtained using a statistical treatment, resulted from the Reynolds averaging.

The closure of the mean equations is based on Boussinesq's (1877) hypothesis of eddy viscosity. For the velocity fluctuation correlation tensor, the Reynolds Stress Tensor, the closure takes the form:

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

where ν_t is the eddy viscosity, κ is the turbulent kinetic energy, δ_{ij} is the delta of Kronecker operand and the over-bars indicate averaged variables. The form adopted in this work to express the eddy viscosity ν_t, as a function of the turbulent kinetic energy κ and its dissipation rate ε, is using the Prandtl - Kolmogorov relation:

$$\nu_t = C_\mu \frac{\kappa^2}{\varepsilon} = \frac{1}{Re_t} \ , \tag{4}$$

where C_μ is a constant of value 0,09. With the adoption of relation (4), the $\kappa - \varepsilon$ turbulence model relation imposes the necessity of two supplementary transport equations to the system of mean equations, destined to evaluation of variables κ and ε. Once defined the closure of the system of mean equations, the direction proposed by Brun (1988) produces the following system of equations:

$$\frac{\partial \overline{u}_i}{\partial x_i} = 0 \ , \tag{5}$$

$$\frac{\partial \overline{u}_i}{\partial t} + \overline{u}_j \frac{\partial \overline{u}_i}{\partial x_j} = -\frac{1}{\rho}\frac{\partial \overline{p^*}}{\partial x_i} + \frac{\partial}{\partial x_j}\left[\left(\frac{1}{Re} + \frac{1}{Re_t}\right)\left(\frac{\partial \overline{u}_i}{\partial x_j} + \frac{\partial \overline{u}_j}{\partial x_i}\right)\right] \ , \tag{6}$$

$$\frac{\partial \kappa}{\partial t} + \overline{u}_i \frac{\partial \kappa}{\partial x_i} = \frac{\partial}{\partial x_i}\left[\left(\frac{1}{Re} + \frac{1}{Re_t\,\sigma_\kappa}\right)\frac{\partial \kappa}{\partial x_i}\right] + \Pi - \varepsilon \ , \tag{7}$$

$$\frac{\partial \varepsilon}{\partial t} + \overline{u}_i \frac{\partial \varepsilon}{\partial x_i} = \frac{\partial}{\partial x_i}\left[\left(\frac{1}{Re} + \frac{1}{Re_t\,\sigma_\varepsilon}\right)\frac{\partial \varepsilon}{\partial x_i}\right] + \frac{\varepsilon}{\kappa}\left(C_{\varepsilon 1}\Pi - C_{\varepsilon 2}\varepsilon\right) \ , \tag{8}$$

where:

$$\Pi = \left[\left(\frac{1}{Re_t}\right)\left(\frac{\partial \overline{u}_i}{\partial x_j} + \frac{\partial \overline{u}_j}{\partial x_i}\right) - \frac{2}{3}\kappa\delta_{ij}\right]\frac{\partial \overline{u}_i}{\partial x_j} \ , \tag{9}$$

$$p^* = \bar{p} + \frac{2}{3}\rho\kappa \ , \tag{10}$$

and the constants of the model are given:

$$C_\mu = 0,09 \ , \ C_{\varepsilon 1} = 1,44 \ , \ C_{\varepsilon 2} = 1,92 \ , \ \sigma_\kappa = 1 \ , \ \sigma_\varepsilon = 1,3 \ . \tag{11}$$

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, capable of properly representing the flow in the inner region of the turbulent boundary layer. There are four velocity laws of the wall implemented on Turbo 2D, solved under a special algorithm of minimum residuals of Fontoura Rodrigues (1990 and 1991), which were adopted in the simulation of the turbulent open-flow over an abrupt hill of Loureiro (2004).

2.2.1 Velocity law of the wall of Mellor(1966)

Deduced from the equation of Prandtl for the boundary layer flow, considering the pressure gradient term for integration, this wall function is a primary approach to flows that suffer influence of adverse pressure gradients. Its equations are, respectively, for the laminar and turbulent region

$$u^+ = y^+ + \frac{1}{2}p^+y^{+2} \quad , \tag{12}$$

$$u^+ = \frac{2}{K}\left(\sqrt{1+p^+y^+} - 1\right) + \frac{1}{K}\left(\frac{4y^+}{2+p^+y^+ + 2\sqrt{1+p^+y^+}}\right) + \xi_{p^+} \quad , \tag{13}$$

where the asterisk super-index indicates dimensionless quantities of velocity u^+, pressure gradient p^+ and distance to the wall y^+, as functions of scaling parameters to the near wall region, K is the Von Karman constant, and ξ_{p^+} is Mellor's (1966)integration constant, function of the near-wall dimensionless pressure gradient, determined in his work of (1966). The relations between the dimensionless near wall properties and the friction velocity u_f are:

$$y^+ = \frac{y\,u_f}{\nu} \quad , \quad u^+ = \frac{\overline{u}_x}{u_f} \quad \text{and} \quad p^+ = \frac{1}{\rho}\frac{\partial \bar{p}}{\partial x}\frac{\nu}{{u_f}^3} \quad . \tag{14}$$

In equation (13) the term ξ_{p^+} is a value obtained from the integration process proposed by Mellor (1966) and is a function of the dimensionless pressure gradient. Its values are obtained through interpolation of those obtained experimentally by Mellor (1966), shown in table (1).

Table 1. Mellor's integration constant (1966)

p^+	$-0,01$	$0,00$	$0,02$	$0,05$	$0,10$	$0,20$	$0,25$	$0,33$	$0,50$	$1,00$	$2,00$	$10,00$
ξ_{p^+}	$4,92$	$4,90$	$4,94$	$5,06$	$5,26$	$5,63$	$5,78$	$6,03$	$6,44$	$7,34$	$8,49$	$12,13$

2.2.2 Velocity law of the wall of Nakayama and Koyama (1984)

In their work, Nakayama and Koyama (1984) proposed a derivation of the mean turbulent kinetic energy equation, that resulted in an expression to evaluate the velocity near solid boundaries. Using experimental results and those obtained by Stratford (1959), the derived equation is

$$u^+ = \frac{1}{K^*}\left[3(t-t_s) + ln\left(\frac{t_s+1}{t_s-1}\frac{t-1}{t+1}\right)\right] \quad , \tag{15}$$

with

$$t = \sqrt{\frac{1+2\tau^*}{3}} \quad , \quad \tau^* = 1 + p^+y^+ \quad , \quad K^* = \frac{0,419 + 0,539p^+}{1+p^+} \quad and \quad y_s^+ = \frac{e^{K\,C}}{1+p^{+(0,34)}} \quad , \tag{16}$$

where K^* is the expression for the Von Karman constant modified by the presence of adverse pressure gradients, τ^* is a dimensionless shear stress, $C = 5,445$ is the log-law constant and t, y_s^+ and t_s, a value of t at position y_s^+, are parameters of the function.

2.2.3 Velocity law of the wall of Cruz and Silva Freire (1998)

Analyzing the asymptotic behavior of the boundary layer flow under adverse pressure gradients, Cruz and Silva Freire (1998) derived an expression for the velocity. The solution of the asymptotic approach is

$$u = \frac{\tau_w}{|\tau_w|}\frac{2}{K}\sqrt{\frac{\tau_w}{\rho} + \frac{1}{\rho}\frac{dp_w}{dx}y} + \frac{\tau_w}{|\tau_w|}\frac{u_f}{K}ln\left(\frac{y}{L_c}\right) \quad with \quad L_c = \frac{\sqrt{\left(\frac{\tau_w}{\rho}\right)^2 + 2\frac{\nu}{\rho}\frac{dp_w}{dx}u_R} - \frac{\tau_w}{\rho}}{\frac{1}{\rho}\frac{dp_w}{dx}} \quad , \tag{17}$$

where the sub-index $_w$ indicates the properties at the wall, K is the Von Karman constant, L_c is a length scale parameter and u_R is a reference velocity, evaluated from the highest positive root of the asymptotic relation:

$$u_R{}^3 - \frac{\tau_w}{\rho} u_R - \frac{\nu}{\rho} \frac{\partial p}{\partial x} = 0 \; . \tag{18}$$

The proposed equation for the velocity (17) has a behavior similar to the log law far from the separation and reattachment points, but close to the adverse pressure gradient, it gradually tends to Stratford's equation (1959). The same process was used to derive the temperature law of the wall by Cruz and Silva Freire (1998).

3. Numerical methodology

The numerical solution of the proposed system of governing equations, has as main difficulties: the coupling between all equations; the non-linear behavior resulting of the simultaneous action of convective and eddy viscosity terms; the explicit calculations of boundary conditions in the solid boundary and the methodology of use the continuity equation as a manner to link the coupling fields of velocity and pressure.

The numerical solution used in this work considers 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 was presented in Soares and Fontoura Rodrigues (2003).

4. Numerical results

The test case of the open-channel turbulent flow over an abrupt hill was simulated with Turbo 2D using the log-law of the wall and the three laws of the wall previously presented, to evaluate the velocity field in the inner region of the boundary layer. The results obtained from those simulations were compared to the experimental data from Loureiro (2004).

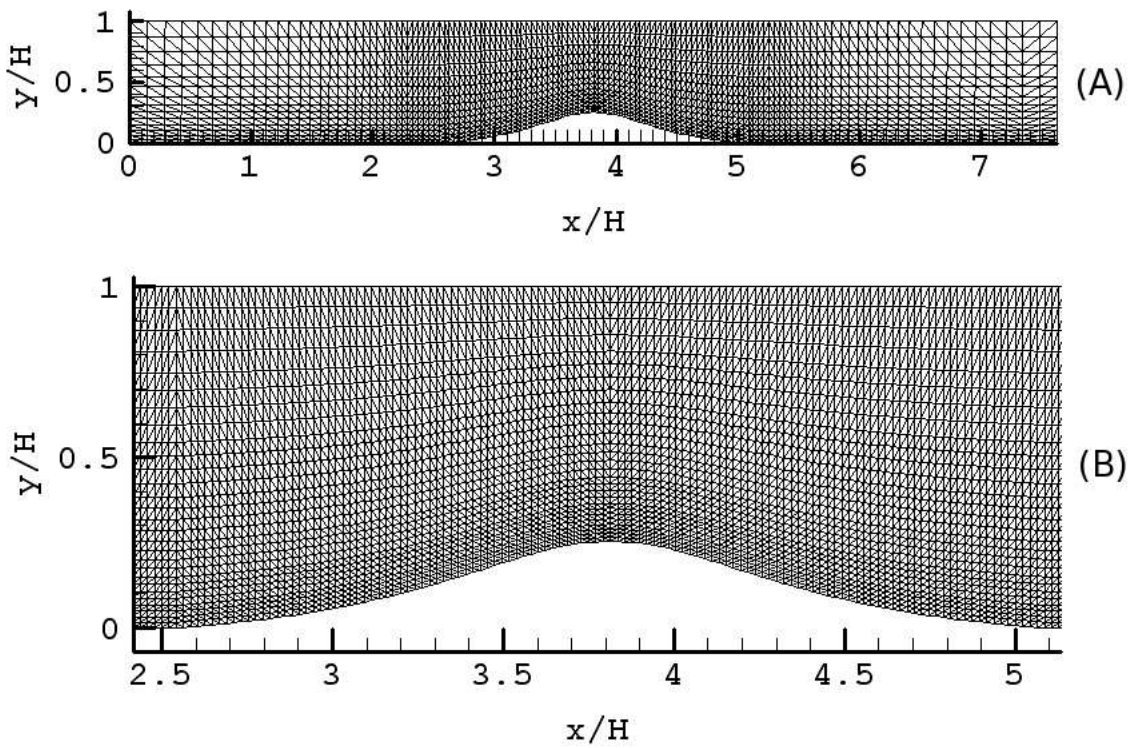

Figure 1. Calculation domain of the test case of Loureiro (2004) discretized in a P1 mesh (A) and a detail of the Iso/P2 mesh around the hill area (B).

The studied experimental test case is characterized by a turbulent open-channel water flow, in a channel with 17 meters of total length and a cross section with 0.6 meters of hight and 0.4 meters wide, with an obstacle defined by a hill situated

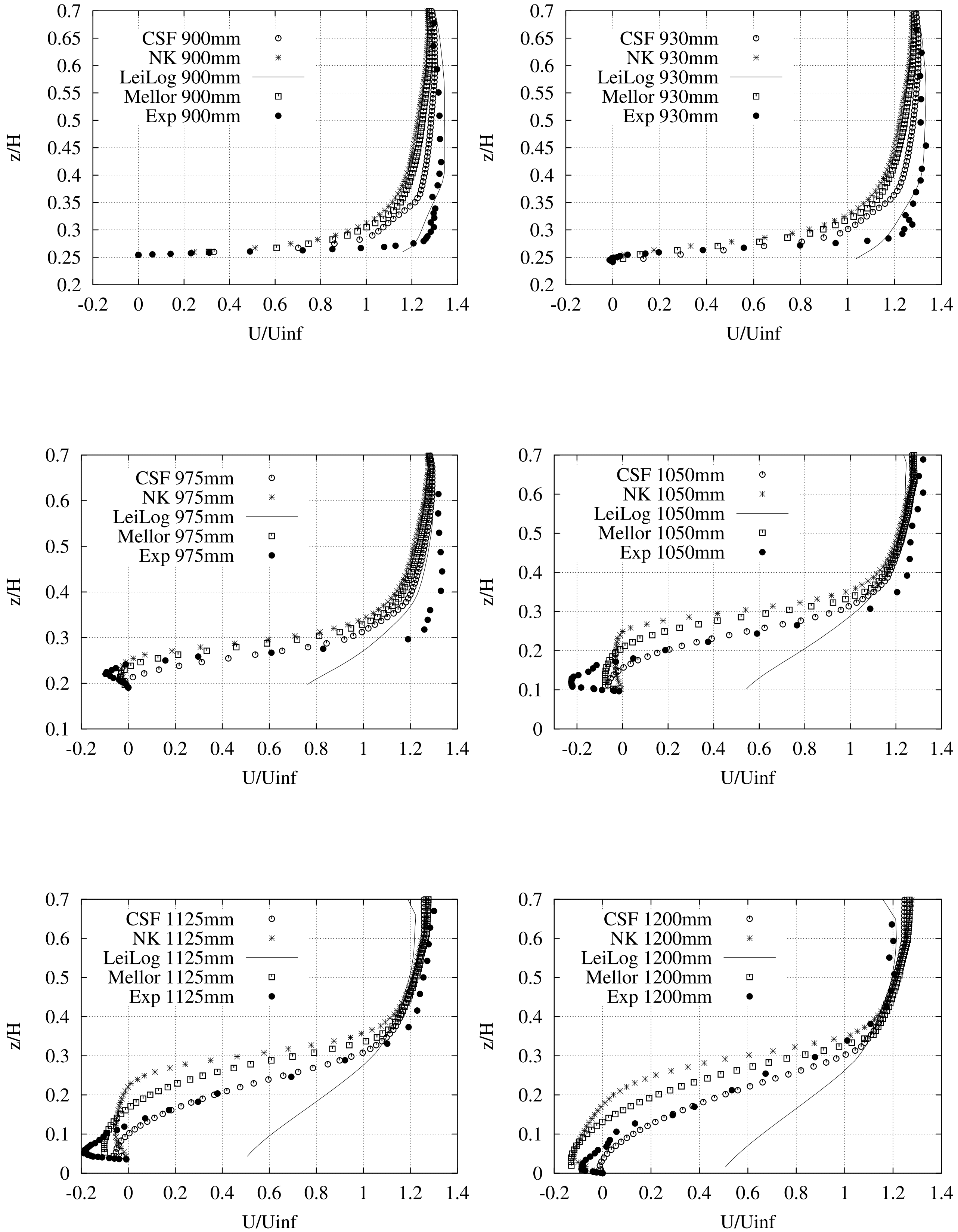

Figure 2. Velocity profiles at six sections - Cruz and Silva Freire (CSF), Nakayama and Koyama (NK), Logarithmic (LeiLog) and Mellor laws of the wall against the experimental data (Exp).

at 8 meters from the beginning of the channel. This hill is an elevation of the bottom of the channel defined by a curve known as the "Witch of Agnesi", given by the relation:

$$Z = \frac{75}{1 + \left(\frac{x^2}{150}\right)^2} - 15 \,, \tag{19}$$

with all dimensions in millimeters, where x and Z are, respectively, the longitudinal and vertical lengths. The flow rate imposed is 4.10^{-3} cubic meters per second, which generates a water depth of 0.236 meters, due to physical restrictions at the end of the channel. The experimental data were taken at $x = 900mm$ $(x/H = 3.81)$, $x = 930mm$ $(x/H = 3.94)$, $x = 975mm$ $(x/H = 4.13)$, $x = 1050mm$ $(x/H = 4.45)$, $x = 1125mm$ $(x/H = 4.76)$ and $x = 1200mm$ $(x/H = 5.08)$.

The finite-elements mesh that discretizes the solution domain covers a region of 1.8 meters of the channel. The base of the hill measures 0.6 meters, and its center is coincident with the center of the mesh. The hight of the mesh is equal to the water line of the experimental case. Figure (1A) shows the finite elements mesh P1 used, with 1875 nodes and 3472 elements, and the complete numerical domain. Figure (1B) shows a detail of the Iso/P2 mesh around the hill, and had 7221 nodes and 13888 elements in total. Based on a water height H, equal to 0.236 meters, the Reynolds number of this case is 10000.

The boundary conditions were set to experimental profiles of velocity and turbulent kinetic energy at the inlet boundary, atmospheric pressure and reference values for all other variables, measured experimentally, at the upper boundary. At the outlet boundary the conditions imposed are atmospheric pressure and null-derivative for all others variables.

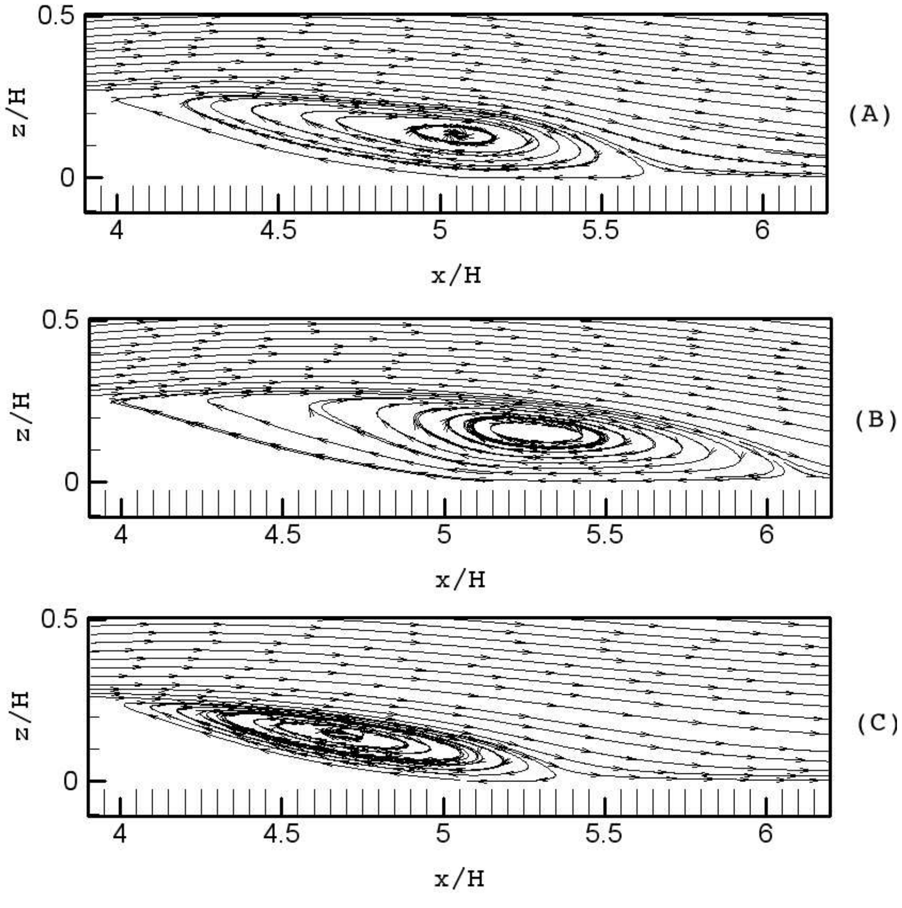

Figure 3. Recirculating zone obtained with the law of the wall of Mellor (A), Nakayama and Koyama (B) and Cruz and Silva Freire (C).

In figure (2), the obtained stream-wise velocity profiles are shown, in six stations of measurements taken from the work of Loureiro (2004). At $x = 900mm$ and $x = 930mm$, the best agreement with the experimental data comes from the log-law simulation. But, in the hill's descent part, where the boundary layer detachment occurs, only the log-law fails completely in predicting the flow characteristics. The best agreement of the other three laws comes from the law of the wall of Cruz and Silva Freire (1998), which predicts more precisely the velocity profiles. This can be specially seen in the

$x = 1050mm$, $x = 1125mm$ and $x = 1200mm$ velocity profiles. The laws of the wall of Mellor (1966) and Nakayama and Koyama (1984) results show less agreement with the experimental data, with latter detachment and reattachment point, as seen in figure (3) and table (2).

Table 2. Results of obtained recirculating zones from the laws of the wall

Turbo 2D **Law of the wall**	**Detachment** $\mathbf{x_d/H}$	**Reattachment** $\mathbf{x_r/H}$	**Recirculating zone** $\mathbf{L = (x_r - x_d)/H}$	**Error** $\mathbf{\lvert L - L_{exp} \rvert}$	**Error %** $\mathbf{\lvert L - L_{exp} \rvert / L_{exp} \times 100}$
Mellor	4.00	5.65	1.65	0.082	5.22%
Nakayama and Koyama	3.95	6.05	2.1	0.532	33.9%
Cruz and Silva Freire	4.05	5.35	1.3	0.268	17.09%
Experimental	**3.94**	**5.50**	**1.568**	——	——

The discussions about the pressure field along the channel are qualitative, comparing the numerical results of the four laws of the wall among them. The experimental results of the pressure field are not yet available.

The numerical results showed four different configurations of the pressure field, but all of them followed the same characteristic: two regions of adverse pressure gradients and two regions of stream- wise pressure gradients, with the highest absolute pressure values in the near-wall region. From the beginning of the numerical domain, the pressure increases gradually, until reaching its maximum value, approximately at 0,75H before the hill. In the small region between the maximum pressure location and the top of the hill, the strongest stream-wise pressure gradient occurs, being the minimum pressure located at the top of the hill. From the top, the pressure starts to increase once again, resulting a second adverse pressure gradient. The pressure almost reaches its maximum value again in a small region, and the detachment of the boundary layer occurs due to this adverse pressure gradient. After this second point of maximum, the pressure starts to decrease once again, until it becomes constant, as it was at the beginning of the domain.

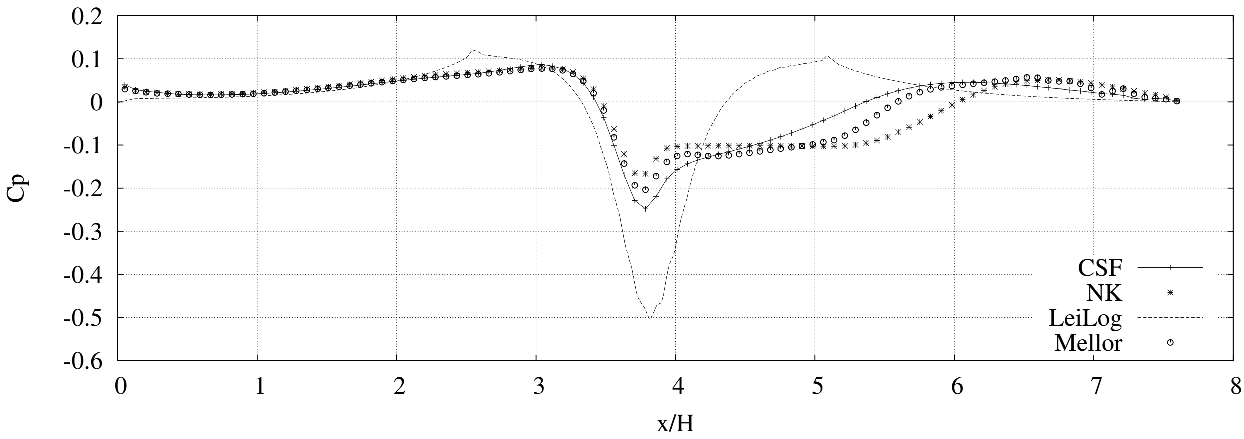

Figure 4. Pressure coefficient (C_P) along the wall for each law of the wall - Cruz and Silva Freire (CSF), Nakayama and Koyama (NK), Logarithmic (LeiLog) and Mellor (Mellor).

In figure (4) it is shown the pressure coefficients along the wall obtained from the simulations with each law of the wall. The result obtained with the log-law clearly shows that the log-law was incapable of predicting the detachment of the boundary layer, while the other three could.

5. Conclusions

To get good and coherent results from numerical simulations of detaching turbulent boundary layer flows, with the use of models present in industrial CFD applications like the two-equation turbulence models proposed by Jones and Launder (1972), is a hard task due to the limitations of this modeling technique, amongst which the most relevant are the linear behavior of Boussinesq's Hypothesis (1877), the supposition of equilibrium between production and dissipation of turbulent kinetic energy and the lack of universality of his constants values.

The use of laws of the wall capable of detecting the effects of adverse pressure gradients, associated with the $\kappa - \varepsilon$ model of Jones and Launder (1972), allow the simulation of the turbulent boundary layer detachment, even under smooth adverse pressure gradients, like occurs in the test case of the asymmetric plane diffuser of Buice and Eaton (1995), as shown the results of Soares and Fontoura Rodrigues (2004, 2003) and Ataídes and Fontoura Rodrigues (2002).

The numerical simulation of the boundary layer detachment of the test case of Loureiro (2004) is particularly hard due to the low intensity adverse pressure gradient that causes the detachment. It is even weaker than the one observed in the test case of Buice and Eaton (1995).

The results obtained show that the best agreement with the experimental data of the longitudinal velocity predictions in the recirculating zone are obtained with the law of the wall of Cruz and Silva Freire (1998), and that the results with the law of Mellor (1966) are closer than those obtained with the law of Nakayama and Koyama (1984). The log law was unable to predict the detachment but provided the best velocity profiles in the region before the hill, before the detachment point.

The equipment used in the simulations was a Pentium IV station, running at 3.0 GHz with 1.0 Gb of RAM. The computational cost necessary to obtain convergence to the final results, in terms of flow time, was 103 seconds with the log law, 511 seconds with Mellor's law (1966), 890 seconds with the law of Cruz and Silva Freire (1998) and 1919 seconds of flow time with Nakayama and Koyama's law (1984), showing that larger computational demand does not mean necessarily an improvement in results.

6. Acknowledgements

We are grateful to Technology and Scientific Enterprize Foundation - FINATEC, for the material and financial support, which made this work possible.

7. References

Ataídes, R.S.C. & Fontoura Rodrigues J.L.A., 2002. "Estudo computacional de escoamentos turbulentos com descolamento de camada limite". 9th Brazilian Congres of Thermal Engineering and Sciences, Caxambu, Brasil.

Brison, J. F., Buffat, M., Jeandel, D. & Serres, D., 1985. "Finite element simulation of turbulent flows, using a two-equation model". Numerical Methods in Laminar and Turbulent Flows, Pineridge Press, Swansea, England.

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

Boussinesq, J., 1877. "Essai sur la théorie des eaux courantes". Mémoires présenté par divers savants à l'Académie des Sciences. Vol. 23, pp. 1-680.

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

Cruz D.O.A. & A.P. Silva Freire. 1998, "On single limits and the asymptotic behaviour of separating turbulent boundary layers". International Journal of heat and Mass transfer. Vol. 41 , no. 14, pp. 2097-2111.

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

Fontoura Rodrigues, J.L.A., 1991. "Um método de mínimo resíduo adaptado ao cálculo de condições de contorno não-lineares no escoamento turbulento bidimensional". XI Congresso Brasileiro de Engenharia Mecânica, pp. 465-468.

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

Loureiro, J.B.R., 2004. "Estudo Experimental de Escoamentos Bidimensionais sobre Orografia". Programa de Bolsas de Alto Nível da União Européia para a América Latina, Tese de Mestrado, Universidade do Porto, Portugal.

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

Nakayama, A. & Koyama, H., 1984. "A wall law for turbulent boundary layers in adverse pressure gradients". AIAA Journal, vol. 22, no. 10, pp. 1386-1389.

Soares, D. V. , Ataídes, R. S. C. & Fontoura Rodrigues, J. L. A., 2003. "Numerical simulation of a turbulent flow over a heated wall". COBEM 2003, 17th International Congress of Mechanical Engineering, November 10-14, 2003, São Paulo, Brasil.

Soares, D. V. & Fontoura Rodrigues, J. L. A., 2004. "The Performance of Thermal and Velocity Wall Laws". ENCIT 2004, 10th Brazillian Congress of Thermal Sciences and Engineering, November 29 - December 03, 2004, Rio de Janeiro, Brasil.

Soares, D. V. & Fontoura Rodrigues, J. L. A., 2004. "Complex Near-Wall Turbulent Flows: A Performance Analisys of Velocity and Temperature Laws of the Wall". ETT 2004, Escola de Primavera de Transição e Turbulência, September 27 - October 01, 2004, Porto Alegre, Brasil.

Stratford, B.S., 1959. "The prediction of separation of turbulent boundary layer". Journal of Fluid mechanics, vol. 5, no. 1, pp 1-16.