A NUMERICAL MODEL OF A THERMAL BOUNDARY LAYER

Gilberto Augusto Amado Moreira, gilberto@cear.ufpb.br
Centro de Energias Alternativas e Renováveis, Universidade Federal da Paraíba, Cidade Universitária - João Pessoa – PB, CEP 58051-900

André Augusto Campagnole dos Santos
Comissão Nacional de Energia Nuclear, Centro de Desenvolvimento da Tecnologia Nuclear, Rua Pr. Mário Werneck s/n - Pampulha - Belo Horizonte – MG, CEP 31123-970

Ramon Molina Valle
Adriana Lessa Viana
Universidade Federal de Minas Gerais, Av. Antônio Carlos, 6627 - Pampulha - Belo Horizonte – MG, CEP 31270-901

Abstract. *The interaction between the atmosphere and the earth's surface influences the momentum exchange processes, heat, water vapor and pollutants, determining what is known as planetary boundary layer (PBL). These processes affect the dynamics and thermodynamic characteristics of the region where they happen. The flow in the PBL exhibit fundamental differences depending on the combination between the flow-through shear forces and thrust forces. The shear master PLC and the neutral stable (night), while the thrust dominates the convective PLC (daytime). The flow patterns and turbulence can be very different depending on which strength prevails. The mechanical codes of computational fluid dynamics (CFD) are not usually used for the simulation of PLC convective, however due to the great detail inherent in this type of tool, especially near the surface, the use of this tool has great potential for this type of simulation. This study aims to determine and evaluate a CFD methodology for the commercial code CFX 16.0, robust and able to simulate the main features of a convective PLC. With this goal was numerically simulated the experiment conducted by Fedorovich et al. in a wind tunnel to a convective PLC evolving horizontally and bounded by a temperature inversion. A mesh sensitivity study was conducted to determine the mesh parameters appropriate for this type of simulation. Four turbulence models of RANS models Family were evaluated: RNGKE (Renormalization Group k-ε), SST (Shear Stress Transport), SSGRS (Speziale-Sarkar-Gatski Reynolds Stress) and BSLRS (Baseline Reynolds Stress). The results showed that the proposed methodology is able to simulate the main qualitative and quantitative characteristics of a convective CLP and its development over a horizontal section. The study indicates that the BSLRS turbulence model, though require more computational effort is the most suitable for the simulation of this type of flow*

Keywords: *Planetary Boundary Layer, Natural Convection, Computational Fluid Mechanics*

1. INTRODUCTION

The planetary boundary layer (PBL) plays an important role in numerous engineering projects such as large buildings, dispersion of pollutants, ventilation, wind power and transmission lines. All of these projects require data collected within the PBL, so know and predict how the PBL will behave in a particular region is critical.

The PBL is characterized by having large temporal and spatial variation of its properties as a function of soil cover, season, solar radiation and influence of man. During the daytime step of the surface of the earth cycle is heated by the sun and heat is transported by air vertically due to buoyancy forces. When the thrust forces are predominant we can sort the PBL as convective. This convective transport results in a thickening of the PLC which can reach 2 km near the equator (Soares, 2004). The flow in the PLC have been studied in three ways: data collection in the field, laboratory experiments and numerical simulations.

Data collection in the field, such as those carried out by Taylor and Teunissen (Taylor et al., 1987) generally provide real data on the PBL in the region studied, but demand great investment and time. Moreover, this type of study usually does not provide high spatial resolution because measurements are made in a very limited number of points.

In experiments performed in the laboratory, such as those conducted in wind tunnels (Fedorovich et al., 1996) and water tanks (Willis et al., 1974), it is possible to isolate certain phenomena and parameters present in the PLC, and by extrapolation, to understand how they act and influence the behavior of a real PLC. These experiments require much time and a large investment, but are very useful, especially for the validation of numerical methods.

The numerical simulations of the PBL can be divided into two categories according to the size of the resolved scale. mesoscale simulations are carried out in large areas comprising hundreds of square kilometers, but its resolution is usually limited near the ground, not providing good results in this (Kristóf et al., 1974) region. As for the micro-scale simulations show areas of only a few kilometers in area and are able to provide high-resolution data near surface region of greatest interest to engineering projects.

Commercial codes of computational fluid mechanics (CFD - abbreviation of the English Computational Fluid Dynamics) as the Ansys CFX 11.0®, (Ansys, 2007), employing the finite volume method on unstructured triangular meshes for the solution of the Navier-Stokes equations mediated Reynolds (RANS - abbreviation of the English Reynolds

Averaged Navier-Stokes) and, in principle, are able to address a PLC micro-scale. These codes are sold as sealed packages, with little or no access to routines and algorithms. Therefore, these codes should be widely validated through comparisons with data collection results in the field and / or experiments conducted in the laboratory before they can be applied to confidence PLC simulations.

The used mathematical modeling in solving the RANS equations is based on the average flow solution and the use of models for the calculation of turbulence, not solving the turbulent scales. In this way the calculations are simplified and demand on the spatial discretization of the domain is reduced. There is a wide range of turbulence models, however, none are suitable for all situations, it is necessary to conduct a study for each new simulation models.

This study aims to evaluate a methodology for the simulation of convective PLC using a commercial CFD code, the Ansys CFX 11.0®, (Ansys, 2007). For this experiment performed by (Fedorovich et al., 1996) modeled in a wind tunnel a convective PBL evolving horizontally limited by a temperature inversion, was simulated and its experimental results compared with numerical. mesh sensitivity studies and turbulence models were performed. The RNGKE models (Renormalization Group k-ε), SST (Shear Stress Transport), SSGRS (Speziale-Sarkar-Gatski Reynolds Stress) and BSLRS (Baseline Reynolds Stress) were evaluated.

2. METHODOLOGY

The RANS equations for mass balance, momentum and energy, as well as the turbulence model and the Boussinesq approximation to the term source of thrust were used to model numerically one convective PBL. The experiment conducted by (Fedorovich et al. 1996) where a convective PBL evolving horizontally limited by an inversion temperature in a wind tunnel, was chosen for validation of numerical results. The choice was mainly because the detail and control of boundary conditions and the quality of the available results. The details of the model, boundary conditions, numerical parameters and mesh are presented in the following sections.

3.1 Model

The modeled area is the wind tunnel cavity specially developed for the simulation of the typical convective conditions in the lower atmosphere, where the convective mixed layer develops on a heated surface and is bounded above by a temperature inversion (Fedorovich et al., 1996). The tunnel is of the type with closed loop test section 10 m long, 1.5 m wide and 1.5 m high. The return tunnel section is divided into 10 layers and each layer is individually inflated, each of which has 15 cm with fan and independent heating system, enabling the prescription of different input profiles. The lower wall (floor) is made of flat aluminum plates through which heat is supplied at a constant flow through a heating system. speed and temperature measurements are carried out in the center tunnel into three stations indicated in Figure. 1 by the letters A, B and C.

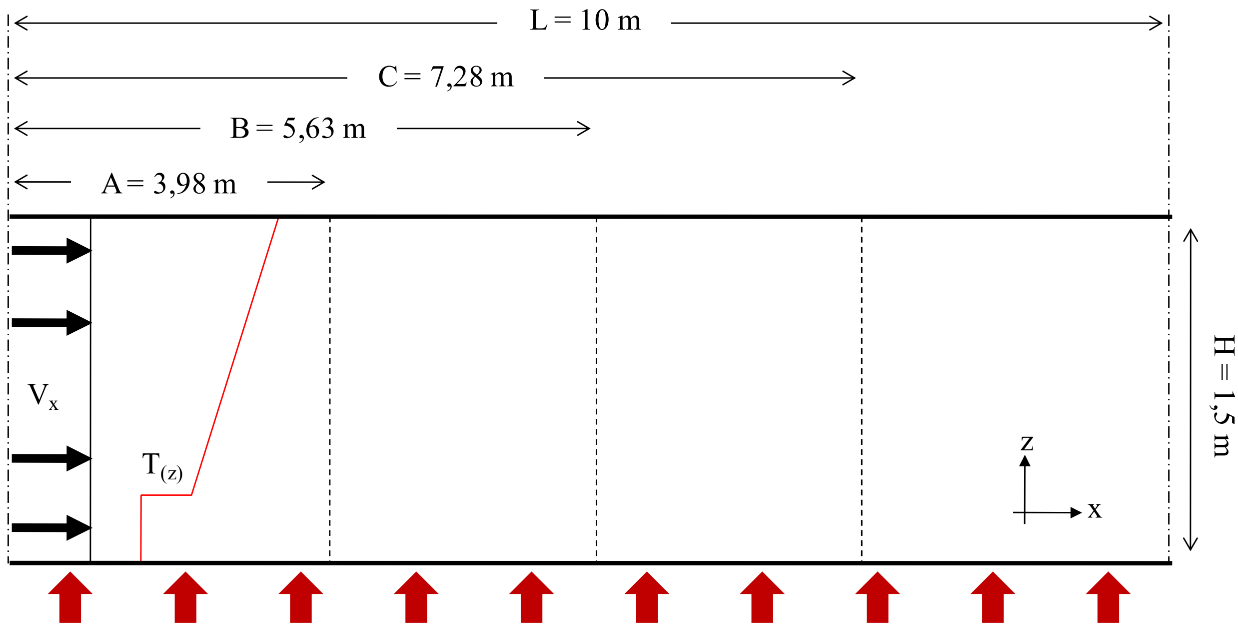

Figure. 1. Computational domain regions of air inflow boundary (black arrows), temperature profile (T (z)) and heat flux input (red arrow).

2.2 Boundary conditions

The conditions of the numerical simulation were configured as the experiment (Fedorovich et al., 1996). A constant heat flux 1250 Wm^{-2} was set on the floor. On entry the computational domain was set a uniform velocity profile to 0.95 *m/s* and a temperature profile with a heat exchange at a height zi equal to 300 mm, as shown in Figure 2.

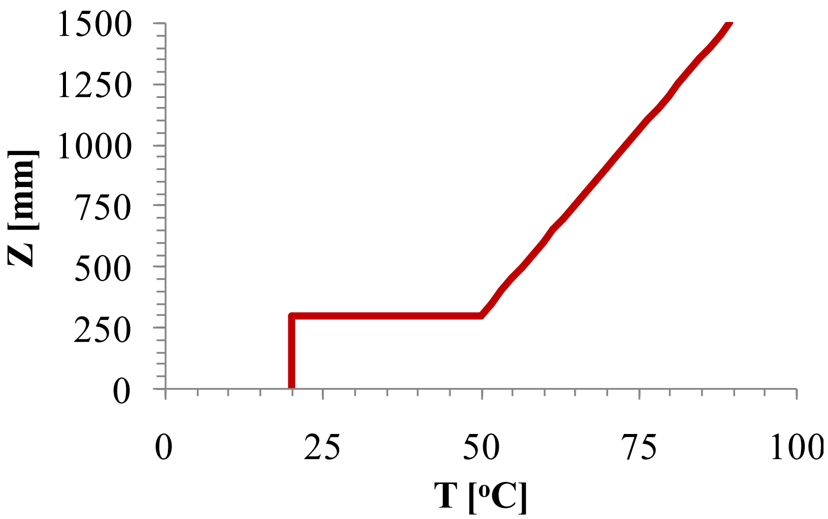

Figure 2. Temperature profile in the wind tunnel entrance.

As the measurements were performed in the experiment (Fedorovich et al., 1996), the numerical simulation was considered in steady state. As the turbulence input is unknown, it assumed an average value of 5%. At the exit of the numeric field is specified on an average pressure condition equal to 0 Pa. The side and top walls were smooth walls and defined as adiabatic.

2.3 Numerical parameters

The numerical simulations were performed using the commercial CFD code, the Ansys CFX 11.0® (Ansys, 2007), which is based on the finite volume method. The RANS equations were discretized using the scheme of differences centered for the diffusive terms and the hybrid scheme of second order for the advective terms of equations. The RMS residual error tolerated for the final convergence was 10-4 for all simulations. The simulations were performed using up to six personal computers, Intel Core Duo 2.8 GHz with 4 GB of RAM, processing in parallel. Other control equations including the continuity, momentum and energy equations which are developed for the whole system can be written as follows.

Continuity equation:

$$\frac{\partial \rho}{\partial t}+\frac{\partial(\rho u)}{\partial x}+\frac{\partial(\rho v)}{\partial y}+\frac{\partial(\rho w)}{\partial z}=0 \tag{1}$$

Momentum equations:

$$\frac{\partial(\rho u)}{\partial t}+\frac{\partial(\rho uu)}{\partial x}+\frac{\partial(\rho vu)}{\partial y}+\frac{\partial(\rho wu)}{\partial z}=-\frac{\partial p}{\partial x}+\mu\left(\frac{\partial^2 u}{\partial x^2}+\frac{\partial^2 u}{\partial y^2}+\frac{\partial^2 u}{\partial z^2}\right), \tag{2}$$

$$\frac{\partial(\rho v)}{\partial t}+\frac{\partial(\rho uv)}{\partial x}+\frac{\partial(\rho vv)}{\partial y}+\frac{\partial(\rho wv)}{\partial z}=-\frac{\partial p}{\partial y}+\mu\left(\frac{\partial^2 v}{\partial x^2}+\frac{\partial^2 v}{\partial y^2}+\frac{\partial^2 v}{\partial z^2}\right), \tag{3}$$

$$\frac{\partial(\rho w)}{\partial t}+\frac{\partial(\rho uw)}{\partial x}+\frac{\partial(\rho vw)}{\partial y}+\frac{\partial(\rho ww)}{\partial z}=\rho g\beta(T-T_\infty)+\mu\left(\frac{\partial^2 w}{\partial x^2}+\frac{\partial^2 w}{\partial y^2}+\frac{\partial^2 w}{\partial z^2}\right) \tag{4}$$

Energy equation:

$$\frac{\partial(\rho cT)}{\partial t}+\frac{\partial(\rho cuT)}{\partial x}+\frac{\partial(\rho cvT)}{\partial y}+\frac{\partial(\rho cwT)}{\partial z}=\lambda\left(\frac{\partial^2 T}{\partial x^2}+\frac{\partial^2 T}{\partial y^2}+\frac{\partial^2 T}{\partial z^2}\right). \tag{5}$$

The turbulence models used in this study were the two models RNGKE and SST equations and models seven SSGRS and BSLRS equations. All models were evaluated using the best network, which was defined through a study of mesh, which used only the BSLRS model because this is the most complex model mathematically between the study.

The turbulence model k-ε (Launder and Spalding, 1974) models the characteristics scales of turbulent viscosity through the turbulent kinetic energy (k) and the turbulent dissipation (ε). The RNGKE (Yakhot and Orzag, 1994) model is an improvement of the previous model, based on the renormalization group of the Navier-Stokes equations. The transport equations for generating turbulence and dissipation are the same k-ε model, differing only in the constants. In Ansys CFX 11.0® (Ansys, 2007), The model uses a scalable RNGKE logarithmic function on the walls, this model limits the lowest value of the wall dimensionless distance used in the calculation of variables.

The SST (Menter, 1994) model involves a combination of turbulence models k-ω (Wilcox, 1994) near the surface, which is based on the k connection with the dissipation rate specifies the turbulence (ω), and k- ε (Launder and Spalding, 1974) core flow. The model includes a function that changes the turbulent viscosity to consider the transport of turbulent shear stresses. In Ansys CFX 11.0® (Ansys, 2007) SST model uses an automatic treatment for the wall function that changes the logarithmic region to the viscous sub-layer according to refining the mesh near the wall.

The Speziale, Sarkar-Gatski Reynolds Stress Model (SSGRS) (Speziale and Sparkar, 1991) solves the six transport equations, one for each turbulent stress (Reynolds stress), and a closure equation for ε. Next to this wall model uses the same treatment described above for RNGKE model in Ansys CFX 11.0® (Ansys, 2007).

The Baseline Reynolds Stress Model (BSLRS) (Menter, 1994) solves, as well as SSGRS, six transport equations for each Reynolds voltage, and solves a closing equation for turbulent dissipation, which is a combination of ε and ω. In Ansys CFX 11.0® (Ansys, 2007). This model uses the same treatment for the next wall region SST model described above.

2.4 Mesh

A study of the sensitivity of results to mesh was conducted evaluating all important parameters of the mesh. On the bottom surface of the area where temperature gradients and speed are greater it is used layers of prismatic elements called "inflation" which are thin in the vertical direction and substantially larger in the longitudinal direction. The use of these elements allows the wall next gradients are obtained without an excessive number of elements required. A superficial control of the element edge is used to concentrate elements near the bottom surface. This control sets the surface element size within a defined time after this point influence the element expands due to an expansion factor, defined as 1.1 for all measured meshes. A small expansion factor ensures that a smooth transition occurs between the elements near the surface and the core of tetrahedral elements defined with 100 mm edge length.

A large number of simulations was performed, Table 1 shows the five knitting parameters that summarize the study. The results of these meshes are presented by the deployment of the mesh parameters that most influence the results. The next mesh wall is evaluated by the number of layers of inflation through the meshes 1 and 2, and the height of the element 1, through the meshes 2 and 3, both parameters of great influence. The distal mesh wall and the spatial resolution of the surface is evaluated through the meshes 4:05 by controlling influence of the surface height and length of the edge surface, respectively. The results for the temperature profiles in position C (shown in Fig. 1) are shown in Figure 3.

Table 1. Parameters of the meshes

Mesh	Inflation		Control on the lower surface		Mesh elements (10^6)	Mesh Node (10^6)
	Layers number	1st layer height (mm)	Edge element (mm)	Height of influence (mm)		
1	20	1	20	100	4,5	1,3
2	30	1	20	100	5,2	1,6
3	30	2	20	100	5,2	1,6
4	30	1	20	300	15,3	3,4
5	30	1	15	100	19,5	5,4

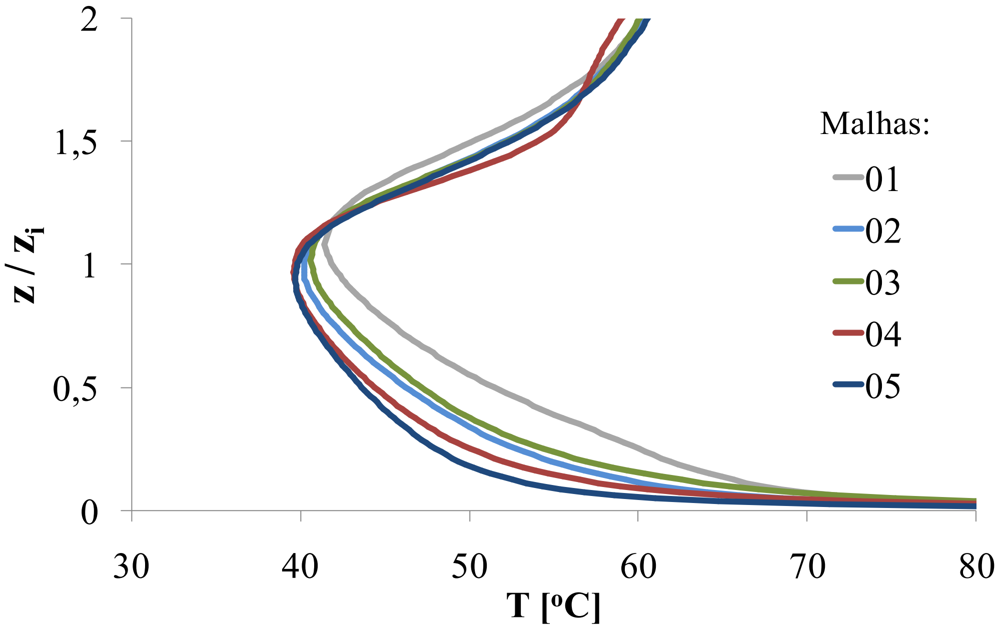

Figure 3. Temperature profile in position C for the various analyzed mesh (7.28 m of input).

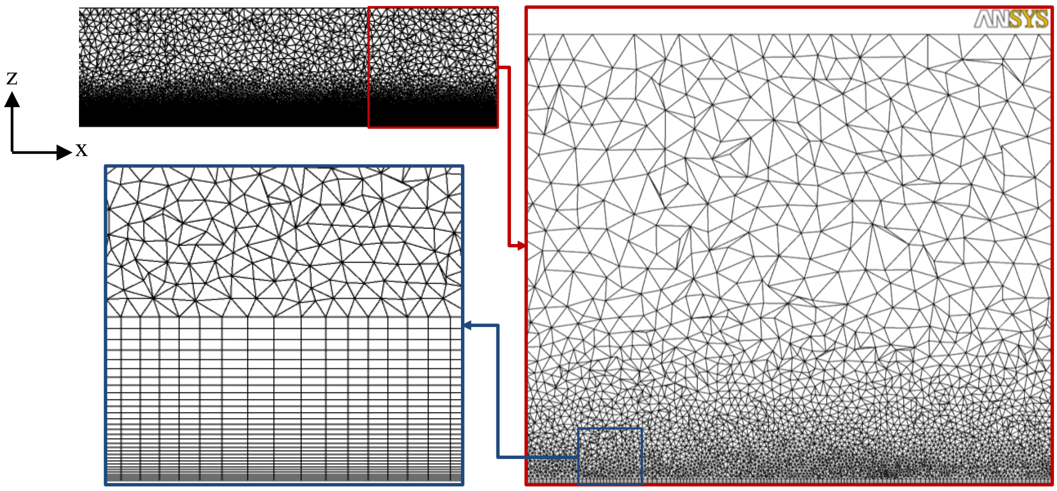

Figure 4. Details of mesh 5 in side view.

From Table 1 it is observed that despite significantly influence the result, inflation mesh parameters little influence on the size of the mesh, as observed for the stitches 1, 2 and 3. Since the parameters of the surface control loop also influence results in significantly, but cause a large increase in the number of elements and nodes of the mesh, as shown in the table meshes 4:05.

Due to the high sensitivity of the mesh results, it is recommended that the mesh used in simulation of convective PBL is the next maximum refined surface and adjacent the region where the higher gradients are present. For this reason we chose to use the mesh 5, shown in Fig. 4 for the other simulations of this study.

3. RESULTS

The results of the temperature profile at the C obtained using the mesh 5 and turbulence models described above are The results of the temperature profile at the C obtained using the mesh 5 and turbulence models described above are compared to the experimental results (EXP) (Fedorovich et al., 1996) in Fig. 5. It is noted that until the ~ 0.5 zi the surface BSLRS model presents the best agreement with the experimental results. On the far region of the wall, above 0.5 zi, the RNGKE model shows better agreement. It is important to note that the turbulence models that use the same wall function like BSLRS and SST and RNGKE and SSGRS showed a very similar behavior, indicating that the next modeling the wall has great influence on the flow behavior in remote areas the walls for this type of flow. Fig. 6 compares the profile dimensionless axial velocity (Vx) and vertical (Vz) obtained from the simulations with the experimental results.

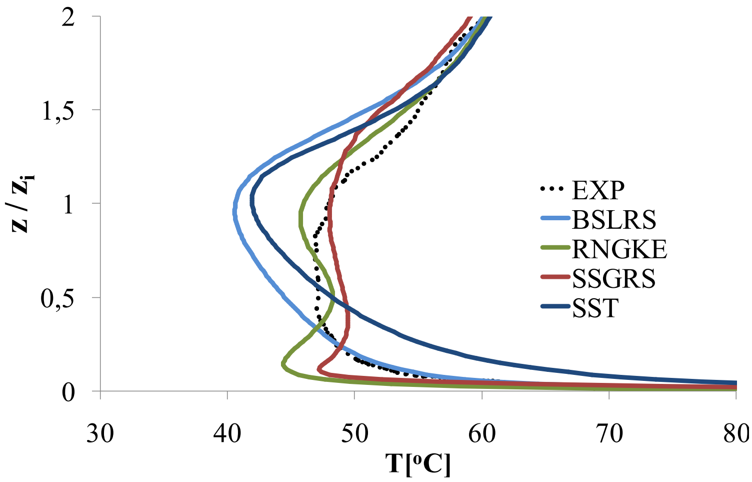

Figure 5. Temperature profiles in C position (7.28 m from the entrance) for different turbulence models.

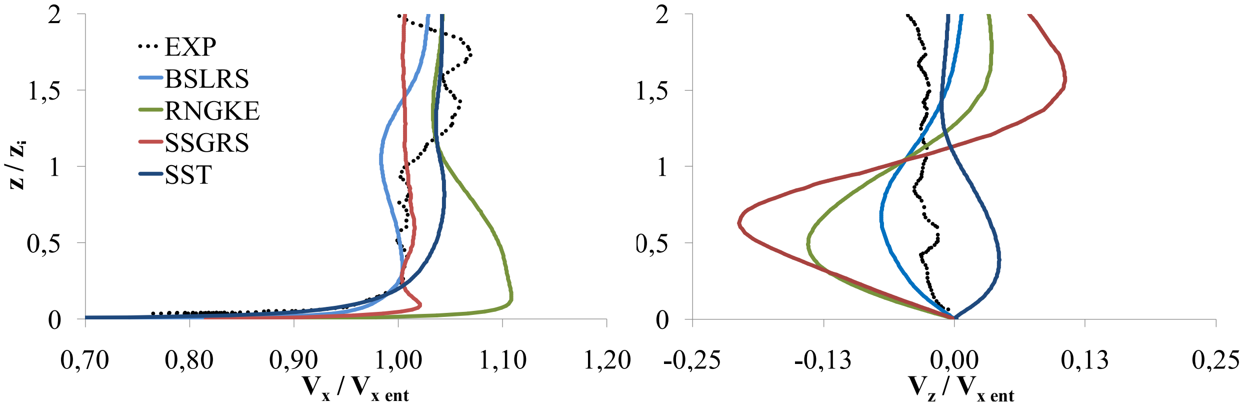

Figure 6. Profiles axial velocity (Vx) and vertical (Vz) at position C for the different turbulence models.

It is again noted that up to 0.5 ~ zi surface BSLRS the model has a better agreement with the experiment, both the axial velocity and to the vertical speed. The axial velocity profile obtained with RNGKE model moves away much of the experimental result, showing an inverse behavior profile relative to experiment.

Fig. 7 shows the temperature contours and the tangential velocity vectors in a front section of the lower portion of the wind tunnel at the C position with all turbulence simulated models. The figure is possible to identify the upside and calculated convective descendants. It is interesting to note that the turbulence models that use a next formulation wall based on ω (BSLRS model and OHS) showed a well organized and less intense convection compared with calculated by the model based on ε near the wall, or the RNGKE and SSGRS. This behavior explains in part the behavior of the temperature profile (Fig. 5) that indicate a higher mixing and RNGKE SSGRS designs in the region near zi.

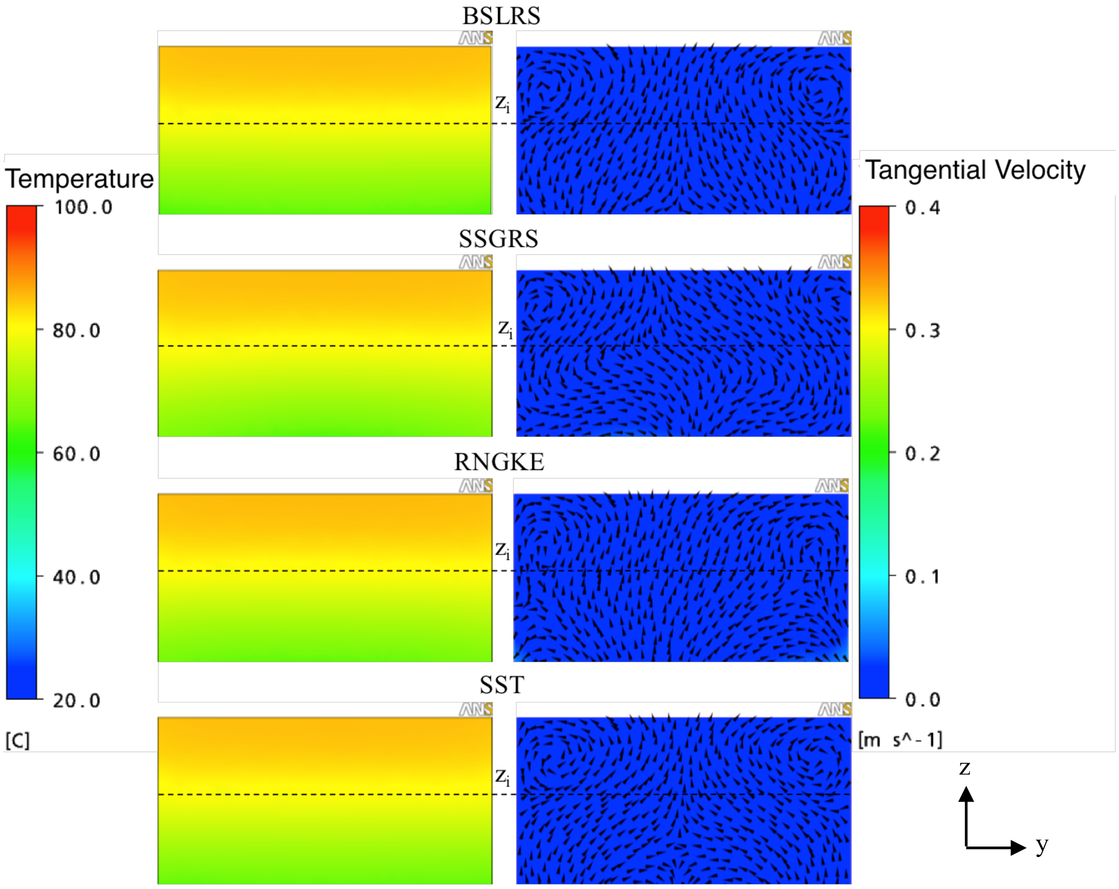

Figure 7. Contours of temperature and tangential velocities in C position (7.28 m from the entrance) to the simulated turbulence models.

4. CONCLUSION

A methodology for the simulation of convective PBL using a commercial CFD code, the Ansys CFX 11.0®, was evaluated. For this experiment performed by (Fedorovich et al. 1996) modeled in a wind tunnel convective PBL evolving horizontally limited by a temperature inversion, it was used for the numerical validation. mesh sensitivity studies and turbulence models were performed. The RNGKE models (Renormalization Group k-ε), SST (Shear Stress Transport), SSGRS (Speziale-Sarkar-Gatski Reynolds Stress) and BSLRS (Baseline Reynolds Stress) were evaluated.

In the mesh sensitivity study noted that although significantly influence the results, the inflation mesh parameters influence the bit mesh size in the order of thousands of elements. Since the parameters of the surface control loop also influence significantly the results, but have a large increase in the number of elements and nodes of the mesh, in the order of millions.

It has been observed that the turbulence models that use a next formulation wall based on ω, ie BSLRS model and OSH, show a well organized and less intense convection compared with calculated by the model based on ε near the wall, ie RNGKE and SSGRS

Whereas in a real convective PBL height zi introduced between 1000 and 2000 m (Soares, 2004), the BSLRS turbulence model proved the most suitable for simulation of this type of flow with respect to engineering problems due to the excellent concordance of temperature profile and speeds of up to 0.5 ~ zi.

It was shown in this study that the methodology proposed using the commercial code Ansys CFX 11.0®, with the BSLRS turbulence model and suitable mesh is able to simulate a convective PBL for engineering purposes.

5. ACKNOWLEDGMENTS

The authors would like to thank the National Council for Scientific and Technological Development (CNPq), projects nº 460258/2014-1

6. REFERENCES (Times New Roman, 10pt, bold, upper-case)

Soares, P. M. M., Parametrização da turbulência e nuvens de camadas limites em modelos atmosféricos, Tese de doutorado, Universidade de Lisboa, 2004.

Taylor, P.A., Teunissen, H.W., The Askervein project: overview and background data, Bound.-Layer Meteor., vol. 39, 15-39, 1987.

Fedorovich, E., Kaiser, R., Rau, M., Plate, E., Wind tunnel study of turbulent flow structure in the convective boundary layer capped by a temperature inversion, J. Atmos. Sci., vol. 53, 1273-1289, 1996.

Willis, G.E., Deardorff, J.W., A laboratory model of the unstable planetary boundary layer, J. Atmos. Sci., vol. 31, 1297-1307, 1974.

Kristóf, G., Rácz, N., Balogh, M., Adaptation of Pressure Based CFD Solvers for Mesoscale Atmospheric Problems, Boundary-Layer Meteorol, vol. 131, 85–103, 2009.

ANSYS CFX 11.0, User manual, ANSYS Europe Ltd., 2007.

Launder, B. E. and Spalding, D. B., The numerical computation of turbulent flow, Computer Methods in Applied Mechanics and Energy, vol. 3, 269-289, 1974.

Yakhot, V., Orzag, S., Renormalization group analysis of turbulence, J. of Sci. Comput., vol. 1, 1-51, 1986.

Menter, F. R., Two-equation eddy-viscosity turbulence models for engineering applications, AIAA-Journal, vol. 32, 269-289, 1994.

Wilcox, D. C., Turbulence modeling for CFD, DCW Industries Inc., La Cañada, Estados Unidos da America, 1994.

Speziale, C. G, Sparkar, S, Gatski, T. B., Modeling the pressure-strain correlation of turbulence: an invariant dynamical system approach, J. Fluid Mechanics, vol. 277, 245-272, 1991.

7. RESPONSIBILITY NOTICE

The following text, properly adapted to the number of authors, must be included in the last section of the paper: The authors is are the only responsible for the printed material included in this paper.