TEST OF TURBULENCE MODELS FOR WIND FLOW ON THE DOWNWIND SLOPE OF A 2D RIDGE MODEL IN NEUTRAL ATMOSPHERE

Marcio Arêdes Martins
Mechanical Enginnering Department, Universidade Federal de Minas Gerais
Av. Antônio Carlos, 6627, Campus Pampulha, Belo Horizonte, MG, Brazil, CEP: 31270-901.
aredes@demec.ufmg.br

Geraldo A. Campolina França
Mechanical Enginnering Department, Universidade Federal de Minas Gerais
Av. Antônio Carlos, 6627, Campus Pampulha, Belo Horizonte, MG, Brazil, CEP: 31270-901.

Ramón Molina Valle
Mechanical Enginnering Department, Universidade Federal de Minas Gerais
Av. Antônio Carlos, 6627, Campus Pampulha, Belo Horizonte, MG, Brazil, CEP: 31270-901.
ramon@vesper.demec.ufmg.br

Abstract. *This work proposes a model for the simulation of a atmospheric flow over topography. The major interest consists on to verify if an atmosferic flow model can be implemeted in the CFX solver to simulate both neutral and stratified conditions. The effects of turbulence models on a neutral atmospheric boundary layer are investigated. Calculations are presented for wind flow over a two-dimensional triangular ridge. Comparisons of predictions with wind tunnel measurements show that both k-ε and RNG-based k-ε models give good agreement with respect to flow profiles and separation length. Compared with the previous studies, the numerical results obtained are very satisfactory in the sense that overall characteristic flows are in agreement to the data reported in the literature. The finite volume method on unstructured and hybrid meshes was employed to solve the equations of the proposed mesoscale model.*

Keywords. *atmospheric boundary layer, turbulence models, wind tunnel, CFX.*

1. Introduction

Almost all scales in respect to space and time can be verified in atmospheric flows. These scales have a strong dependence on a several climate elements such as turbulence, buoyancy, topography effects and rotation of the Earth. Knowledge of the regional distribution of meteorological variables such as wind velocity, temperature, humidity and turbulent kinetic energy is necessary in industrial planning and pollution control. Flow separation, recirculation and local high-speed flows occur when wind flows over valleys and mountains. It is important to consider the location and size of flow separation and recirculation zones in industrial planning, since pollution can be trapped within populated valley. Knowledge of the distribution of wind velocity and its magnitude are crucial features for the design of electrical transmission lines and towers.

A high Reynolds number and the absence of viscous sublayer characterize the flow in a real atmosphere. Thus, it is impossible to avoid using the so-called wall-functions to model the flow near the rough surface. Several micro and mesoscale models to predict wind flows over three-dimensional topologies are presented in literature (Huser et al., 1997; Montavon, 1998; Uchida and Ohya, 1999; Kim and Patel, 2000). The major differences among the abovementioned models are the turbulence models, the assumptions about the relationship of buoyancy and gravitational forces and the boundary conditions.

In this work, the Reynolds averaged Navier-Stokes with wall-functions and two-equation turbulence models are solved on a hybrid unstructured mesh using the finite volume discretization scheme. A general-purpose Navier-Stokes solver (CFX, 2001) was used to solve the equations of the proposed model. The turbulence models compared in this work are the standard k-ε model (Jones and Launder, 1972) and the RNG-based k-ε model (Yakhot and Orszag, 1986). The results of the proposed model are validated using the experimental results of Lee and Park (1998) for the flow over a two-dimensional triangular ridge. Their experiments have been conducted in a wind tunnel where the inflow turbulence intensity and scale length could be controlled. The length of the predicted reattachment behind the ridge is compared to the experimental results and other numerical predictions available in literature.

2. Numerical method

For a neutral atmospheric boundary layer, the flow is assumed isothermal, incompressible and turbulent. The governing equations are the Reynolds averaged Navier-Stokes equations coupled to a turbulent model usually described by a couple of partial differential equations (i.e. k-ε model). The continuity and momentum equations are:

$$\frac{\partial U_j}{\partial x_j} = 0 \, , \tag{1}$$

$$\frac{\partial(U_i U_j)}{\partial x_j} = -\frac{1}{\rho}\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j}\left(\nu\frac{\partial U_i}{\partial x_j} - \overline{u_i u_j}\right), \tag{2}$$

where U_i and u_i are the mean and turbulent fluctuation velocities, respectively, p is the pressure, ρ is the density and ν is the kinematic viscosity. The Reynolds stress ($-\overline{u_i u_j}$) is defined as (Jones and Launder, 1972):

$$-\overline{u_i u_j} = 2\nu_t S_{ij} - \frac{2}{3}k\delta_{ij}, \tag{3}$$

where k is the kinetic energy. The eddy viscosity (ν_t) and the mean strain-rate tensor (S_{ij}) are given by:

$$\nu_t = C_\mu \frac{k^2}{\varepsilon}, \tag{4a}$$

$$S_{ij} = \frac{1}{2}\left(\frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i}\right), \tag{4b}$$

where C_μ is a characteristic constant and ε is the rate of dissipation of turbulent kinetic energy (or just dissipation rate). The mathematical statement of the eddy viscosity defines the so-called turbulence model. In this work, the accuracy of two-equation models k-ε (Jones and Launder, 1972) and RNG-based k-ε (Yakhot and Orszag, 1986) are compared. The coupled equations for these models are given by:

$$\frac{\partial(U_i k)}{\partial x_i} = \frac{\partial}{\partial x_i}\left(\frac{\nu_t}{\sigma_k}\frac{\partial k}{\partial x_i}\right) - \overline{u_i u_j}S_{ij} - \varepsilon, \tag{5a}$$

$$\frac{\partial(U_i \varepsilon)}{\partial x_i} = \frac{\partial}{\partial x_i}\left(\frac{\nu_t}{\sigma_\varepsilon}\frac{\partial \varepsilon}{\partial x_i}\right) - \frac{\varepsilon}{k}\left(C_1 \overline{u_i u_j}S_{ij} + C_2\varepsilon\right), \tag{5b}$$

where the values of the characteristics constants C_μ, C_1, C_2, σ_k and σ_ε define the turbulence models. These constants are given in Tab. (1).

Table 1. Values for the constants for two-equation turbulence models.

Model	C_μ	C_1	C_2	σ_k	σ_ε
k-ε	0.09	1.44	1.92	1	1.3
RNG k-ε	0.085	$1.12 - \frac{\eta(1-\eta/4.38)}{1+0.015\eta^3}$	1.68	0.7179	0.7179

For the RNG-based k-ε model, the parameter η is defined as $\eta = \sqrt{2}S_{ij}k/\varepsilon$ (Yakhot and Orszag, 1986). As already mentioned, the two turbulence models in this study uses the so-called wall functions to describe to flow near the surface. These functions are given by:

$$U = \frac{u_*}{\kappa}ln\left(\frac{z}{z_0}\right), \tag{6a}$$

$$k = u_*^2 C_\mu^{-1/2}, \tag{6b}$$

$$\varepsilon = \frac{u_*^3}{\kappa z}, \tag{6c}$$

where u_* is the friction velocity, z is the vertical distance front the surface, z_0 is the average roughness length and κ is the von Kármán constant (κ= 0.41).

Although most atmospheric flows are typically unsteady, the assumption of steady flow is justified by a slow development in the atmospheric conditions under a short time period. Further assumptions to the flow equations are that no Coriolis force is included and that the buoyancy term is neglected. In the present study, the major interest is in the flow behavior near the ground. Thus, the force due to the ground effects is assumed to dominate the Coriolis force. It is assumed that the atmosphere is neutral or at least weakly stratified, so the vertical momentum equation causes no wind due to the density stratification. The buoyancy effect may be significant when heavy cold or humid air is draining out from a higher altitude location. The numerical model is completed by specifying appropriate boundary conditions.

The surface roughness and stability class are parameters that are set to specify the state of the atmosphere. These parameters are set on the surface and at the inlet. The boundary conditions for the wind velocity (U_i), kinetic energy (k) and dissipation rate (ε) are given by specified profiles at inlet. The atmosphere stability is defined by enforcing these Dirichlet boundary conditions. The inlet profiles are given by (van Ulden and Holtslag, 1985; Duynkerke, 1988):

$$U = \frac{u_*}{\kappa}\left(ln\left(\frac{z}{z_0}\right) - \psi_M\left(\frac{z}{L}\right) + \psi_M\left(\frac{z_0}{L}\right)\right), \tag{7}$$

$$k = \frac{u_*^2}{\sqrt{C_\mu}}\left(1-\frac{z}{h}\right)^2, \tag{8}$$

$$\varepsilon = \frac{u_*^3}{\kappa}\left(\frac{1}{z}+\frac{4}{L}\right), \tag{9}$$

where L is the mean Obukov length for neutral (L > 10 km) or stable atmosphere, h is the boundary layer depth and the correction function Ψ_M is given by:

$$\psi_M(z) = -17\left(1-e^{-0,29\frac{z}{L}}\right). \tag{10}$$

The form of the correction function $\psi_M(z)$ was used in a similar study by (van Ulden and Holtslag, 1985). The friction velocity (u_*) is calculated by using a reference velocity U_{ref} at an altitude z_{ref} in Eq. (7). The boundary layer depth is calculated using the equation proposed by Duynkerke (1988):

$$h = 0.4\sqrt{\frac{u_* L}{f}}, \tag{11}$$

where f is the Coriolis parameter. For stable atmosphere, the mean Obukov length can vary from 60 meters to 10 kilometers (Duynkerke, 1988). It is important to remark that Eqs. (7) to (10) are used in the literature to simulate stable condition. These boundary conditions are implemented in the CFX solver in order to simulate the stable flow in further studies. However, these boundary conditions can be also used to simulate the stable flow condition by setting the Obukov length grater the 10 km (van Ulden and Holtslag, 1985). In this study, a typical value of L = 130 km is used in this study.

The wall boundary condition is applied at the no slipping surface, which is considered rough. The boundary condition at outlet is set by specifying the pressure and setting homogeneous Neumann boundary condition for the other variables.

The CFX solver employs a finite volume discretization scheme on unstructured and hybrid mesh, mainly formed by triangles and quadrilaterals. The basic discretization scheme adopted in CFX is a conventional UDS with numerical advection correction (NAC) for the advection terms in the momentum equations. NAC improves the accuracy of the UDS scheme by including a blending term in the discretization. The pressure-velocity coupling in the mass and momentum equations is handled by the introduction of a fourth order "pressure redistribution term" in the discretized equations to overcome the problem of checkerboard oscillations, which are found when the variables are collocated. The method is similar to that used by Rhie and Chow (1983) with a number of extensions, which improves the robustness of the discretization when the pressure varies rapidly, or is affected by body forces. Further details can be found in CFX (2001).

3. Simulation of wind flow over a triangular ridge

Arya and Shipman (1981) and Lee and Park (1988) were presented experimental results of the wind flow over a triangular ridge. Their experiments were performed in a closed-return type subsonic wind tunnel having 1.8 m wide × 1.5 m high × 11 m long test section. Several vortex generators and roughness elements were installed in front the ridge. These turbulence elements are able to generate a thermally neutral atmospheric boundary layer. The experimental uncertainty reported by Arya and Shipman (1981) was about 15%. Further details related to the experimental apparatus are given by Lee and Park (1988). The calculation domain and the unstructured mesh are presented in Fig. (1a), and the ridge dimension and details of the hybrid mesh are presented in Fig. (1b).

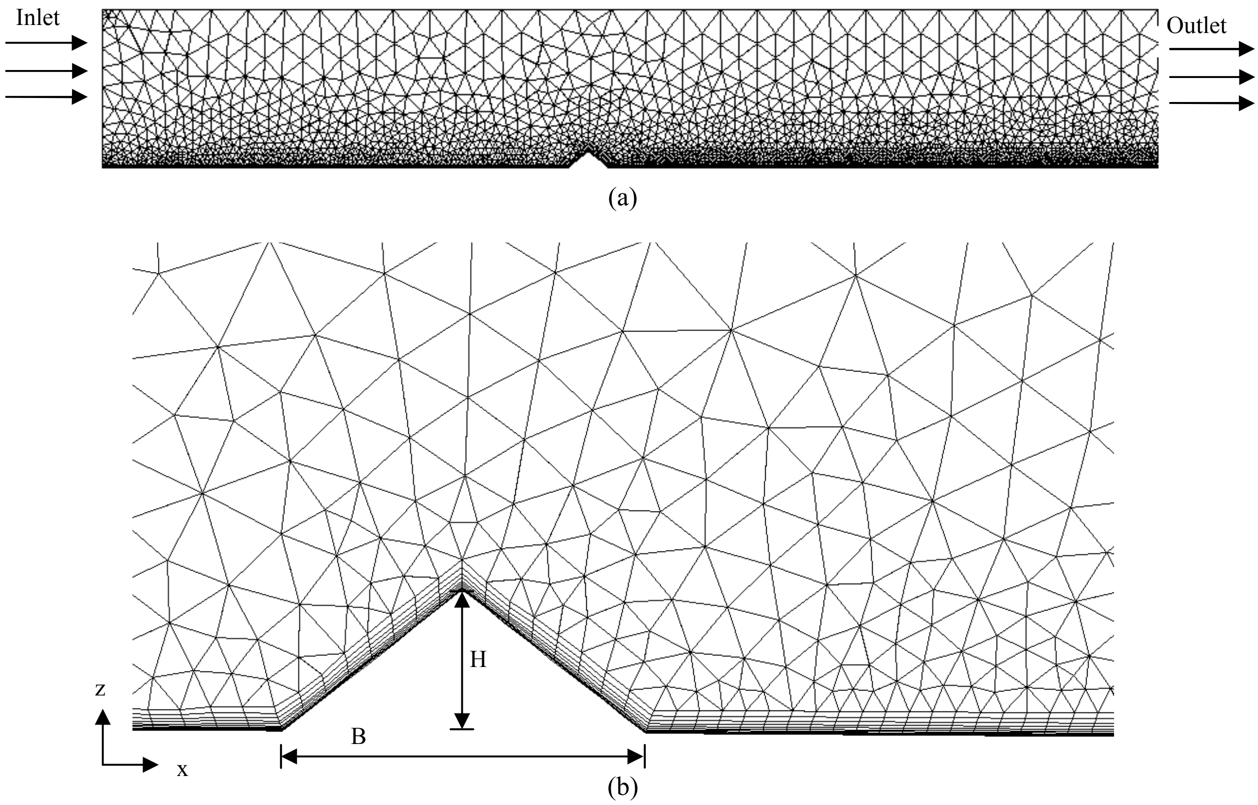


Figure 1. (a) Calculation domain and unstructured mesh, and (b) triangular ridge dimensions.

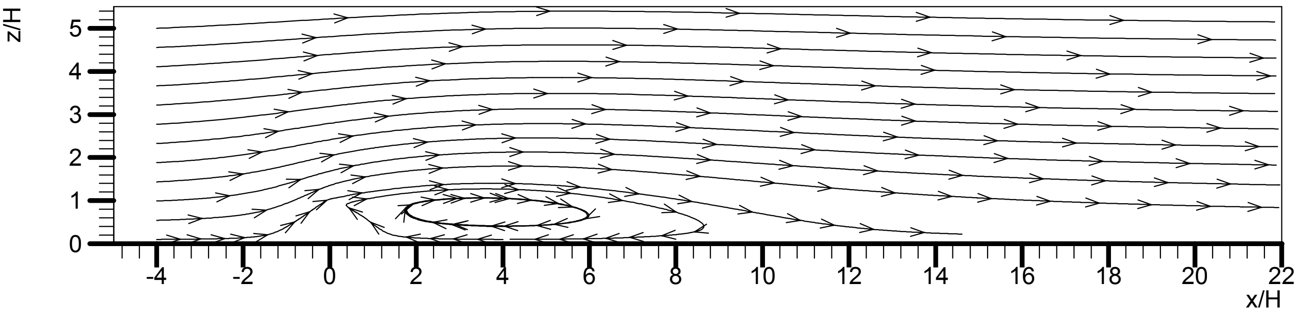
Figure (1a) shows that mesh refinement was applied near the surface, where higher gradients of wind velocity are expected. Figure (1b) shows the geometric parameter of the triangular ridge: height (H = 0,15 m) and base length (B = 0,385 m). Hence, it can be verified in Fig. (1b) that a quadrilateral mesh was used. This body-fitted mesh improves the numerical solution near the wall. The mesh used in this work is formed by 8403 triangles and 4573 quadrilaterals, which correspond to 12976 polyhedral and 5190 nodes.

Inflow profiles were set at inlet (Fig. (1a)) using Eq. (7) to Eq. (9). The experimental value for the airflow velocity at z_{ref} = 0.15 m was fixed by Arya and Shipman (1981) to U_{ref} = 14 m/s. Turbulence models were tested using the same boundary conditions. The streamlines for the standard and RNG turbulence models are presented in Figs. (2a) and (2b), respectively.

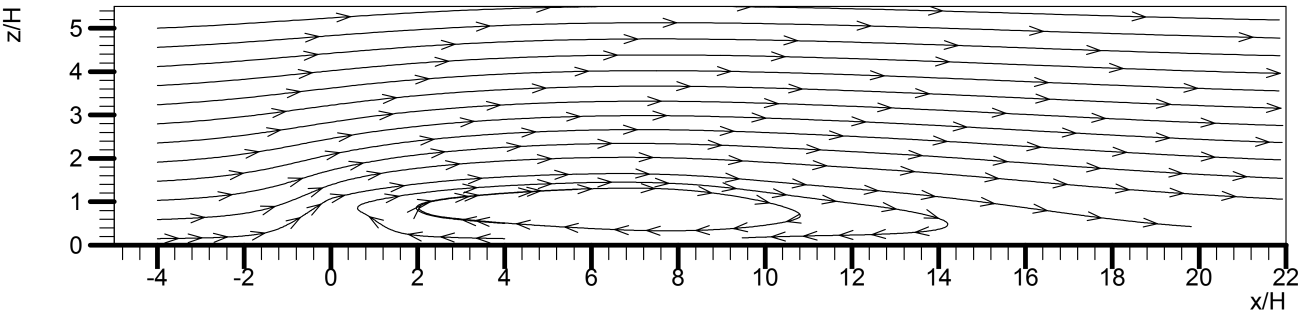
The coordinate system was changed as a function of the triangular ridge height (H). The origin was defined as the base of the ridge, in which *(x, z) = (0, H)* is the vertex of the ridge. Figures (2a) and (2b) show that the recirculation zones for the k-ε model and RNG k-ε are solely different in length. The recirculation length was calculated from the center of the ridge, as a function of the ridge height. A comparison of the predicted recirculation length to those presented in literature is presented in Tab. (2).

Table 2. Recirculation length behind the triangular ridge.

Model / Experimental measure	Recirculation length
Experiment (Arya and Shipman, 1981)	13.0*H*
Modified k-e (Mouzakis and Bergeles, 1991)	10.0*H*
k-ε (This study)	9.3*H*
RNG k-ε (This study)	15.6*H*



(a)



(b)

Figure 2. Streamlines for wind flows using (a) k-ε and (b) RNG k-ε models.

The RNG turbulence model predicts the recirculation length closest to the wind-tunnel measure (Tab. (2)). A predicted value presented by Mouzakis and Bergeles (1991) is close to the present value (9.3*H*). These authors presented a modified k-ε model, in which $C_\mu = 0.033$ and wall functions including pressure gradients. Mouzakis and Bergeles (1991) have justified the changing on the C_μ constant in order to correct the over-estimation of the eddy viscosity, in the upper atmosphere where the wind shear is weaker. Figures (3a), (3b) and (3c) present the vertical profiles of the horizontal velocity.

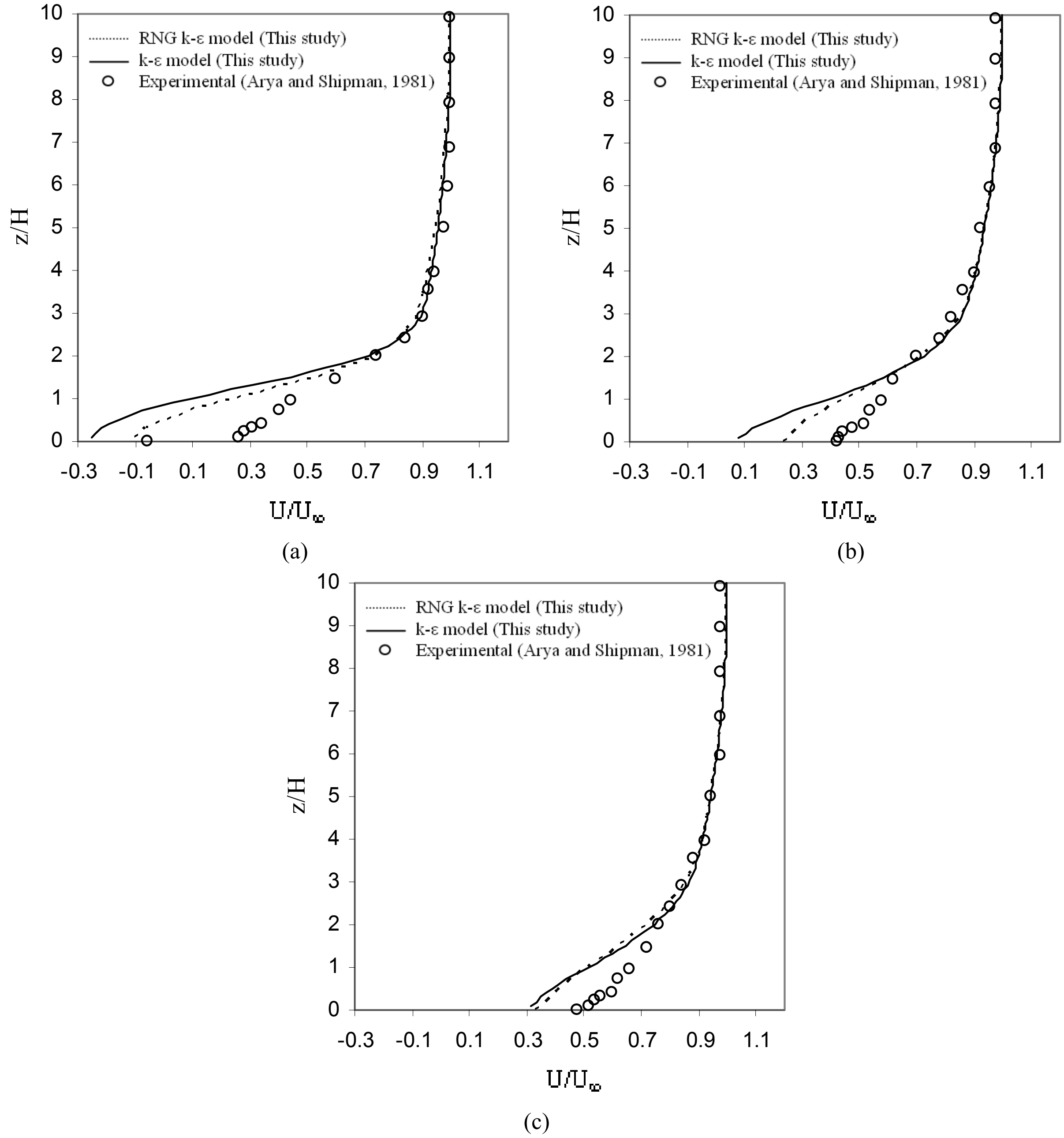


Figure 3. Horizontal velocity profiles behind a ridge at (a) x/H = 8, (b) x/H = 19 and (c) x/H = 29.

Figure 3 shows good agreement to the experimental measures of horizontal velocity, except in the separation region at x/H = 8 nearest the surface. Measures in this region are suspect due to the experimental apparatus used by Arya and Shipman (1981) was not able to determine the flow direction. Other researches (Mousakis and Bergeles, 1991; Jung, 1994) have found similar differences in their computational results at this region. The predictions using the RNG k-ε turbulence model are in better agreement to the experimental measurements, when compared to the standard k-ε model.

The k-ε model is widely used for numerical studies of atmospheric boundary layer (see Montavon, 1998; Uchida and Ohya, 1999), in which researches have been verified that this model is suitable for wind field predictions, since the C_μ were changed *to* 0.033.

4. Conclusions

Two-equation turbulence models with wall functions were used to simulate the flow over a triangular ridge. The proposed boundary conditions include the effects of atmospheric stability and roughness of the surface. These conditions can be used to simulate both neutral and stable atmospheric conditions, since suitable correction functions for the velocity profile and mean Obukov length be used. The numerical results obtained using the CFX solver was verified using experimental measures available for a wind tunnel for neutral conditions. The comparison to experimental data shows that the RNG k-ε model is more suitable to predict the wind flow behind the ridge, when compared to the standard k-ε model. From this study, it could be verified that the CFX solver can be use to simulate the atmospheric flow over topology. In further studies, it will be verified if the C_μ constant for the standard k-ε model can be changed. It will be verified if different wall functions can be implemented in the CFX solver since more reliable functions exist to model the flow behind the ridge.

5. References

Arya and Shipman, 1981, "An experimental investigation of flow and diffusion in the distribuited boundary layer over a ridge - I. Mean flow and turbulence structure", Atmosphe and Environment, Vol. 15, pp. 1173-1184.

CFX, 2001, "CFX-5 Solver and Solver Manager Guide", AEA Technology Engineering Software Ltd, UK.

Duynkerke, P.G., 1988, "Application of k-e turbulence closured model to the neutral and stable atmospheric boundary layer", Journal of Atmospheric Science, Vol. 45, no. 5, pp. 390-406.

Huser, A., Nilsen, P.J., and Skatun, H., 1997, "Aplication of k-ε model to the stable ABL: pollution in complex terrain", Journal of Wind Engineering and Industrial Aerodynamics, Vol. 67-68, pp. 425-436.

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

Jung, S.J., 1994, "Application of k-ε turbulence numerical model to a flow and dispersion around triangular ridge (I)", Journal of Korea Air Pollution Reserch, Vol. 10, pp. 116-123.

Kin, H.G., and Patel, V.C., 2000, "Test of turbulence models for wind flow over terrain with separation and recirculation", Boundary-Layer Metereology, Vol. 94, pp. 5-21.

Lee, S.J., and Park, C.W., 1998, "Surface preassure variations on a triangular prism by porous fences in a simulated atmospheric boundary layer", Journal of Wind Engineering and Industrial Aerodynamics, Vol. 73, pp. 45-58.

Montavon, C., 1998, "Validation of a non-hydrostatic numerical model to simulate startified wind fields over complex topography ", Journal of Wind Engineering and Industrial Aerodynamics, Vol. 74-76, pp. 273-282.

Mouzakis, F.N., and Bergeles, G.C., 1991, "Numerical prediction of turbulence flow over a two-dimensional ridge", International Journal of Numerical Methos in Fluids, Vol. 12, pp. 279-296.

Rhie, C.M., and Chow, W.L., 1983, "Numerical study of turbulence flow past an airfloil with trailing edge separation", AIAA Journal, Vol. 21, pp. 1527-1532.

Uchida, T. and Onya, Y, 1999, "Numerical Simulation of Atmospheric Flow Over Complex Terrain", Journal of Wind Engineering and Industrial Aerodynamics, Vol. 81, pp. 283-293.

van Ulden, A.P., and Holtslag, A.A.M., 1985, "Estimation of atmospheric boundary layer parameters for diffusion applications", Journal of Clime and Applied Metereology, Vol. 24, pp. 1196-1207.

Yakhot, V., and Orszag, S.A., 1986, "renormalization Group of analysis of turbulence. I. Basic theory", Journal of Scientific Computations, Vol. 1, pp. 3-51.