Wind and Structures, Vol. 13, No. 6 (2010) 487-498
DOI: http://dx.doi.org/10.12989/was.2010.13.6.487

Numerical calculation of the wind action on buildings using Eurocode 1 atmospheric boundary layer velocity profiles

M.F.P. Lopes*[1], J.M. Paixão Conde[1,2], M. Glória Gomes[3] and J.G. Ferreira[3]

[1]*IDMEC, Instituto Superior Técnico, Universidade Técnica de Lisboa, Av. Rovisco Pais, 1049-001 Lisboa, Portugal*
[2]*Departamento de Eng. Mecânica e Industrial, Faculdade de Ciências e Tecnologia, Universidade Nova de Lisboa, 2829-516 Caparica, Portugal*
[3]*DECivil/ICIST, Instituto Superior Técnico, Universidade Técnica de Lisboa, Av. Rovisco Pais, 1049-001 Lisboa, Portugal*

(Received March 26, 2009, Accepted January 11, 2010)

Abstract. When designing structures to the wind action, the variation of the mean wind velocity and turbulence parameters with the height above the ground must be taken into account. This paper presents the numerical simulation results of atmospheric boundary layer (ABL) airflows, in a numerical domain with no obstacles and with a cubic building. The results of the flow characterization, obtained with the FLUENT CFD code were performed using the $k-\varepsilon$ turbulence model with the MMK modification. The mean velocity and turbulence intensity profiles in the inflow boundary were defined in accordance with the Eurocode 1.4, for different conditions of aerodynamic roughness. The maintenance of the velocity and turbulence characteristics along the domain were evaluated in an empty domain for uniform incident flow and the ABL Eurocode velocity profiles. The pressure coefficients on a cubic building were calculated using these inflow conditions.

Keywords: atmospheric boundary layer; wind action on buildings; CFD simulations; Eurocode 1.4.

1. Introduction

In the design of the aerodynamic behaviour of a building or structure, there are several levels of insight which can be considered. Both the static forces and the dynamic effects caused by the wind action can be relevant to the design of the structure. For the calculation of these effects in specific cases where the guidelines of the Eurocode part 1.4 (EC1 2004) are not considered to be sufficient, the choice between wind tunnel testing and numerical calculations using Computational Fluid Dynamics (CFD) codes is still not straightforward (Stathopoulos 1999).

The main difficulties in calculating the wind action on buildings using CFD are (Huang *et al.* 2007): high Reynolds number; impinging at the front face; sharp edges of bluff bodies and effect of the wake in the outflow boundary. In the present case, where it is also aimed to analyse the effect of

* Corresponding Author, Researcher, E-mail: mlopes@hidro1.ist.utl.pt

the incident velocity profile, the problem of the maintenance of the Atmospheric Boundary Layer (ABL) velocity profile is also conditioning.

In order to define the wind action on buildings, it is crucial to define properly the characteristics of the incident mean velocity and turbulence profiles. This definition is frequently not precise, due to the non uniform conditions of aerodynamic roughness, caused by other buildings and the topographic characteristics (Li *et al.* 2009).

To take into consideration the effect of the neighbouring structures and simplify the design process, the variation of the mean velocity and turbulence with height is considered to be represented by profiles that are only dependent on height and a roughness parameter.

In the EC1, the mean velocity profiles are defined by means of a logarithmic formulation

$$\overline{U}(z) = \overline{U}_b 0.19\left(\frac{z_0}{0.05}\right)^{0.07} ln\left(\frac{z}{z_0}\right) \tag{1}$$

Here, $\overline{U}_b$ is the basic wind velocity, characteristic of each geographic area; z is the height above the ground and z_0 is the aerodynamic roughness length. For a height lower than $z_{\min}$, the mean velocity is considered constant, with a value equal to the $z_{\min}$ velocity. This aims to take into account the large variability of the wind velocity in the first meters above the ground.

Eq. (1) can be related with the classical definition of the logarithmic profile through a parameter d, constant for each profile

$$d = \overline{U}_b 0.19\left(\frac{z_0}{0.05}\right)^{0.07} = \frac{u^*}{K} \tag{2}$$

where, u^* is the friction velocity and K is the Von Karman constant (approximately equal to 0.4).

The EC1 defines five terrain categories, according to their aerodynamic roughness, corresponding to the parameters represented in Table 1.

The turbulence intensity, defined as the ratio between the standard deviation and the mean wind velocity, is imposed accordingly with the EC1 as

$$I_u(z) = \frac{1}{ln(z/z_0)} \tag{3}$$

Table 1 Terrain categories according to EC1

Terrain category	z_0 [m]	$z_{\min}$ [m]
0 – Sea or coastal area exposed to open sea	0.003	1
I – Lakes or flat and horizontal area with negligible vegetation and without obstacles	0.01	1
II – Area with low vegetation such as grass and isolated obstacles (trees, buildings) with separations of at least 20 obstacle heights	0.05	2
III – Area with regular cover of vegetation or buildings or with isolated obstacles with separations of maximum 20 obstacle heights	0.30	5
IV – Area in which at least 15% of the surface is covered with buildings and their average height exceeds 15 m	1.00	10

The values of k (turbulent kinetic energy) and ε (dissipation rate of the turbulent kinetic energy) are usually obtained through the following expressions

$$k = \frac{3}{2}(\bar{U}(z).I_u(z))^2 \tag{4}$$

$$\varepsilon = \frac{d^3K^2}{z} \tag{5}$$

As a consequence of their definitions, the EC1 ABL profiles correspond to constant turbulent kinetic energy profiles, only dependent on the terrain roughness.

In this work, two groups of cases were analyzed. In the first group, only the upstream part of the numerical domain was used, without any obstacles. This was done to evaluate how an ABL velocity and turbulence distributions can be effectively maintained from the inflow boundary to the building. In a second stage, the flow in the complete domain including the building was calculated using as inflow conditions either a uniform flow or the EC1 velocity and turbulence profiles.

2. Model definition

2.1 Mesh and domain specification

The application case under analysis is a 30 m side cubic building. The following simplifications were assumed in the geometry: (a) the building has sharp eaves; (b) the building's walls are flat and without roughness elements (windows, balconies); (c) the aerodynamic effects of the surrounding buildings are completely represented by the ABL profiles; (d) the wind direction is perpendicular to one of the building walls.

The numerical domain was defined according to the following dimensions relative to the cube's side L: in the flow direction *22 L*, being *9 L* upstream and *12 L* downstream the building; the domain's height was defined as *8 L* and the width *10 L*. The criteria for the specification of these dimensions were: in the inflow, up and side boundaries of the domain, to avoid obstacle influenced static pressure distributions on these boundaries; in the outflow, to avoid the limitation of the wake's length to influence the correct representation of the flow. The initial upstream domain length was 6 L (as suggested in Huang *et al.* 2007), but it was verified that the distribution of static pressure in the inlet boundary was significantly influenced by the presence of the obstacle. This was considered to be incompatible with the fundamental EC1 definition of the ABL profiles and, as a consequence, the upstream domain length was changed to 9 L, which resulted in constant static pressure on the inlet boundary.

The mesh was generated with two blocks, taking into account the type of flow. In the zone around the cube, in a volume with dimensions $3\,L \times 3\,L \times 2\,L$. (length × width × height), a structured mesh was defined with growing elements starting from the cube's walls. The thickness of the first element near the walls (0.02 cm) was defined in order to accomplish the condition of having the values of the non-dimensional parameter y^+ (see Eq. (11)) in the range 30-300. Outside this inner domain an unstructured mesh was built, being finer near the floor and at the center of the domain, to account for the larger velocity gradients in these zones. Each of the building's faces was discretized by 25 × 25 elements, more refined near the cube's edges. The final mesh was made of 1.2×10^6 elements.

2.2 Boundary conditions

The mean velocity, k and ε profiles were defined in the inflow boundary according to Eqs. (1) to (5) and Table 1 through a 'User Defined Function' (UDF) (Fluent 6.2 User's Guide 2005). The lateral and top boundaries of the domain were defined as solid walls with zero shear stress. At the building walls a *no slip* condition was imposed with the standard wall functions.

The application of the boundary condition on the floor needed a special care, in order to maintain an ABL profile along the numerical domain.

By definition, an ABL turbulence and velocity profile is in equilibrium with a uniformly distributed aerodynamic roughness on the floor. According to Blocken *et al.* (2007), four conditions are necessary to accomplish the equilibrium of an ABL flow type through an empty domain in a commercial code: a sufficiently fine mesh in the vertical direction near the floor of the computational domain; horizontal homogeneous ABL in the upstream and downstream region of the domain; a distance from the centre point of the cell adjacent to the bottom wall larger than the physical roughness height; the knowledge of the relation between the sand-grain type roughness and the correspondent roughness height. However, it is not possible in the standard commercial codes to fulfill all the conditions, because of the definition of the wall functions (that are usually not-changeable).

By computing a complete code using the model of Richards and Hoxey (1993) however, it is possible to achieve the stability of the ABL in a very long domain, being this model not applicable in commercial codes. In FLUENT, it is feasible to change the value of the floor's roughness. This is made using the 'Standard Wall Functions' and changing the parameters 'Roughness height' ε_R and 'Roughness constant' C_s. The value of ε_R is different from z_0 (roughness length). These values are related by (see demonstration in Blocken *et al.* (2007))

$$\varepsilon_R = z_0 \frac{E}{C_s} \tag{6}$$

where E is a empirical constant whose value is 9.793 in FLUENT. Using the standard value for C_s of 0.5, the following is obtained

$$\varepsilon_R = 20 z_0 \tag{7}$$

2.3 Turbulence model

The choice of the adequate turbulence model is essential to solve the air flow around a building effectively. In this work, the standard $k - \varepsilon$ and $k - \varepsilon$ with MMK modification were the turbulence models used. The need for modifying the standard model arises from the excessive turbulent kinetic energy near the obstacle when this model is used in bluff bodies (Huang *et al.* 2007).

The modification known as MMK was first presented in Tsuchiya *et al.* (1998). This modification changes the calculation of the eddy viscosity v_t to reduce the production of turbulent kinetic energy around the body. The eddy viscosity is the parameter that models the moment transfer due to turbulent eddies. Having in S and Ω, respectively, the strain and vorticity invariants

$$S^2 = \frac{1}{2}\left(\frac{du_i}{dx_j} + \frac{du_j}{dx_i}\right)^2 \tag{8}$$

$$\Omega^2 = \frac{1}{2}\left(\frac{du_i}{dx_j} - \frac{du_j}{dx_i}\right)^2 \tag{9}$$

the following modification is introduced

$$\nu_t = C_t^* \frac{k^2}{\varepsilon} \quad \text{with} \quad C_t^* = C_\mu \frac{\Omega}{S} \quad \text{if} \quad \frac{\Omega}{S} < 1 \quad \text{or} \quad C_t^* = C_\mu \quad \text{if} \quad \frac{\Omega}{S} \geq 1 \tag{10}$$

This means, in practice, the reduction of the turbulence viscosity in the zones where $S > \Omega$. In this work, this was applied by choosing the standard $k-\varepsilon$ turbulence model and writing an UDF following the methodology proposed in Huang *et al.* (2007) to change the calculation of the eddy viscosity.

3. Results

3.1 Verification of the ABL stability along an empty domain

In order to confirm that the ABL characteristic mean velocity and turbulent kinetic energy profiles are not significantly changed after flowing through an empty numerical domain, this problem was first addressed during this study.

Fig. 1 presents the results of the verification of the mean velocity profile maintenance along the domain that would later be upstream the building. The results were obtained using a velocity profile correspondent to the terrain type III of EC1 and a base velocity of 10 m/s, changing the floor boundary condition. The roughness was set to 6 m using Eq. (7) and Table 1. The results of the simulations carried out in the FLUENT CFD code show that the velocity variation relatively to the inflow velocity profile can be significant for zones near the floor. It is also verified that adding a roughness condition in the floor boundary contributes significantly to the stability of the profile. In

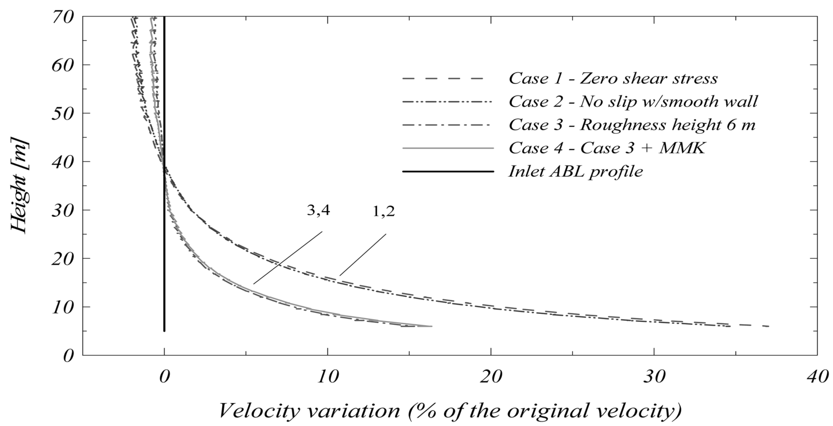

Fig. 1 Verification of the outflow velocity after a 270 m numerical domain, relatively to the inflow velocity profile type III of EC1 (only represented above $z_{\min} = 5$ m) with several floor boundary conditions and $\overline{U}_b = 10$ m/s

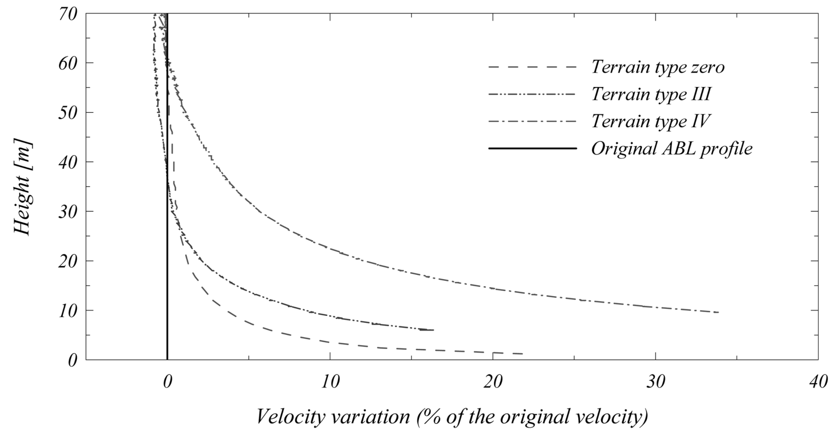

Fig. 2 Velocity variation at the end of the numerical domain, for Case 4 of Fig. 1, changing the inflow velocity profile – profiles for terrain types zero, III and IV are represented

the outer region of the ABL, there is a slight increase of velocity, which is a consequence of momentum balance. As verified from the numerical data, this corresponds to an ascendant flow, especially significant in the beginning of the domain.

Fig. 2 presents the maintenance of the velocity profiles for the same numerical domain, but for different inflow ABL velocity profiles. From the obtained data it can be noticed that the difficulty of maintaining the inflow profile is more significant for profiles with higher aerodynamic roughness. This can be due to the inability of the sand-grain rough wall conditions in FLUENT to represent large roughness conditions, as discussed extensively in Blocken *et al.* (2007) and Hargreaves and Wright (2007).

The maintenance of the inflow turbulent kinetic energy profile was also covered in this study and is represented in Fig. 3. Due to the rough wall definition in FLUENT, that implies a peak in the turbulent kinetic energy dissipation in the cell adjacent to the walls, the maintenance of the turbulent

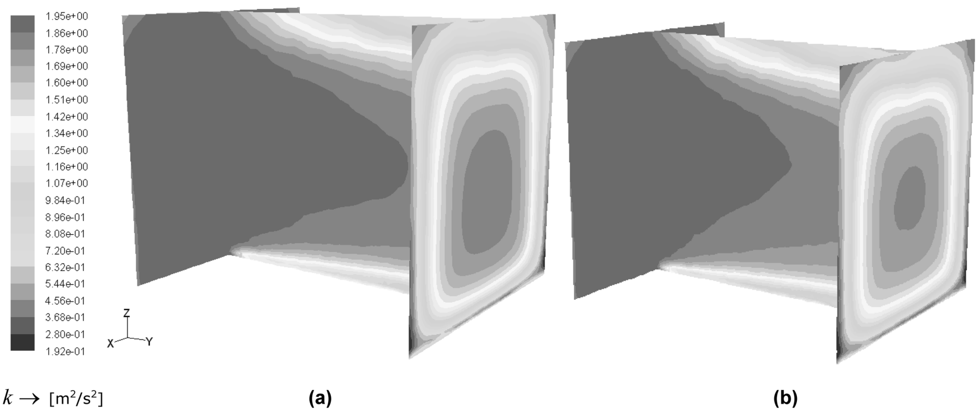

Fig. 3 Variation of the turbulent kinetic energy k along the numerical domain. Inflow constant k distribution correspondent to terrain type III of EC1. Flow is in the positive yy axis direction. The inflow and outflow planes are represented, as well as the longitudinal mean plane. (a) Standard $k-\varepsilon$ and (b) $k-\varepsilon$ + MMK

kinetic energy is not well obtained. The case (a) in Fig. 3, where the standard $k - \varepsilon$ is used, was the one where best results were accomplished. In (b), where the MMK was added to the turbulence model, there is a higher variation of k, which may be related to the fact that the turbulent kinetic energy is limited when adding this alteration.

3.2 Verification of y^+ values

The representation of the non-dimensional parameter y^+ is used to check the correct application of the wall function in the first layer of volumes near the wall, and is defined as

$$y^+ = \frac{\rho u^* y_P}{\mu} \tag{11}$$

Here, ρ and μ are respectively the density and kinematic viscosity of the fluid, u^* is the friction velocity and y_P is the distance of the center in the first volume next to the wall.

The value of the thickness of the first element was adjusted in order to obtain the condition $30 < y^+ < 300$, and a value of 2 cm was used for the first element thickness. Fig. 4 shows the values of y^+ on the center of the first layer of volumes around the cube. It can be observed that, with the exception of a small area in the front face, the indicative range is respected.

3.3 Pressure coefficients for uniform and ABL flows

The pressure coefficient distribution was determined for uniform and ABL flows. This coefficient is given by the following expression

$$C_p = \frac{p}{1/2 \rho v^2} \tag{12}$$

where p is the dynamic pressure at the surface point and v is the flow velocity at an undisturbed

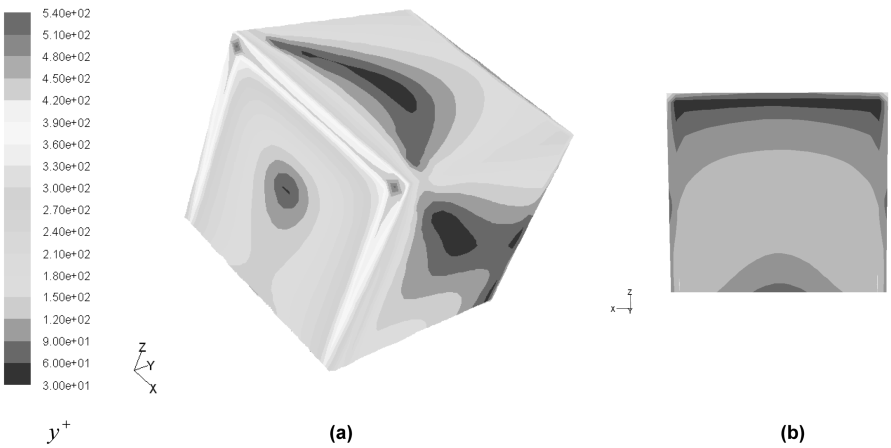

Fig. 4 Values of y^+ on the center of the first layer of volumes that surround the cube. The inflow corresponds to positive velocity in the yy direction. (a) windward, roof and side faces and (b) leeward face

domain point at the obstacle height.

From the analysis of Figs. 5 and 6, where the incident flow is uniform, it is noticeable that in the windward face the results are close to the experimental results by Castro and Robbins (1977) and the difference between the two turbulence models is small. On the roof and the side faces, a significant deviation is verified between the experimental and numerical results near the separation eaves. The negative pressure is excessive in the zone near the eaves, but lower than the experimental values from half of the separation faces downstream. The complete C_p distribution for this case is represented in Fig. 7.

The pressure coefficient distribution obtained for an incident profile correspondent to the EC1 ABL terrain type velocity profiles is shown in Figs. 8 and 9. The results are compared with the experimental values by Stathopoulos and Dumitrescu-Brulotte (1985), obtained for an incident urban profile very similar to the EC1 terrain type III. It can be noticed that the MMK modification is essential to solve the cases where the incident turbulence is high, as it is the case of the ABL-type flows. This becomes clear in the windward face as the values of C_p are higher than one in this face without the MMK modification, which is physically unrealistic. Although the distribution in the

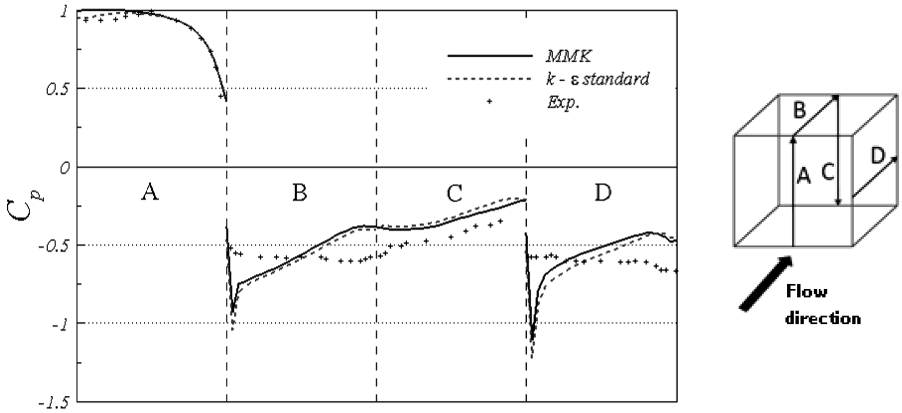

Fig. 5 Pressure coefficients on the central alignments of the faces for uniform incident flow. Comparison between the models with and without the MMK modification and experimental results by Castro and Robbins (1977)

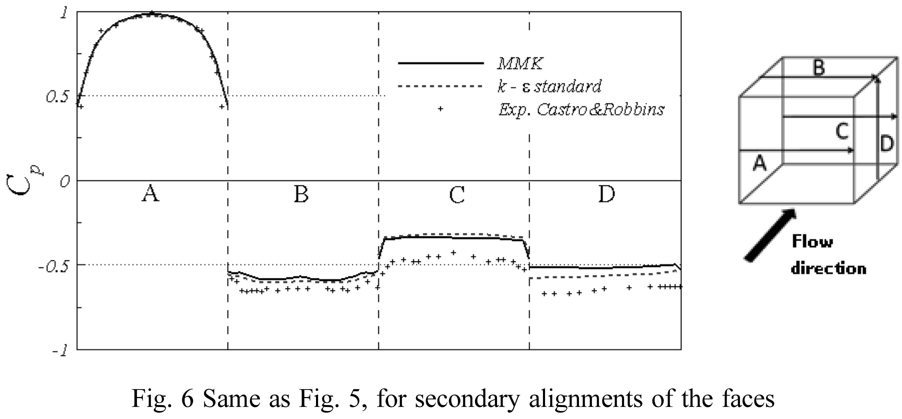

Fig. 6 Same as Fig. 5, for secondary alignments of the faces

central alignment of the front faces is satisfactory, the values near the side eaves and near the floor are slightly higher than expected. This may take place due to the excessive velocity near the floor reported in Fig. 2. The pressure distribution, in the suction faces near the separation eaves, does not represent with precision the target values, as in the uniform flow case. This fact is also reported in Murakami *et al*. (1992) and Tsuchiya *et al*. (1998) and can be justified by the

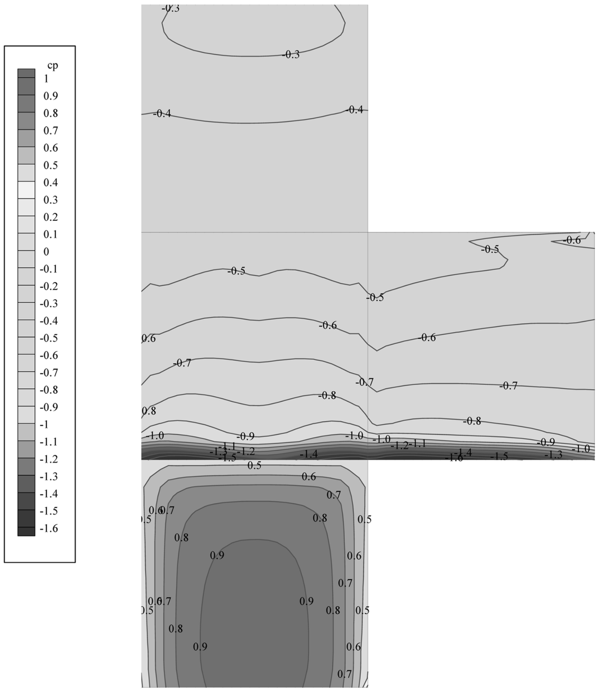

Fig. 7 C_p values on the faces of the cube for uniform incident flow with the MMK modification of the turbulence model

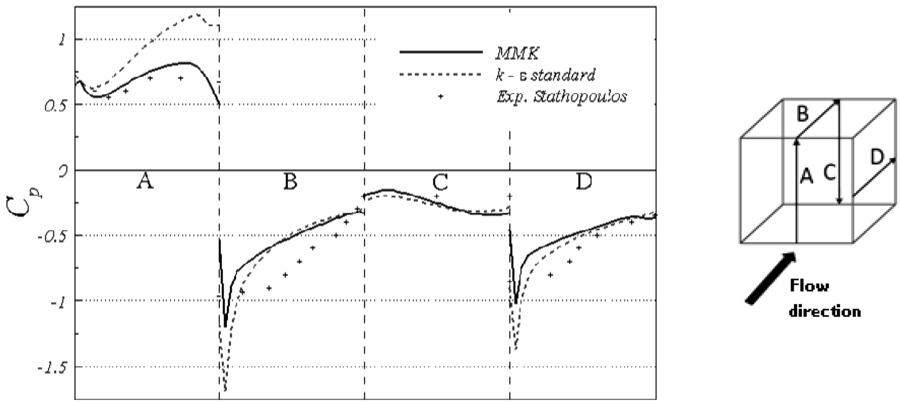

Fig. 8 Pressure coefficients on the central alignment of the faces. Comparison between the simulations with and without the MMK modification and experimental results by Stathopoulos and Dumitrescu-Brulotte (1985)

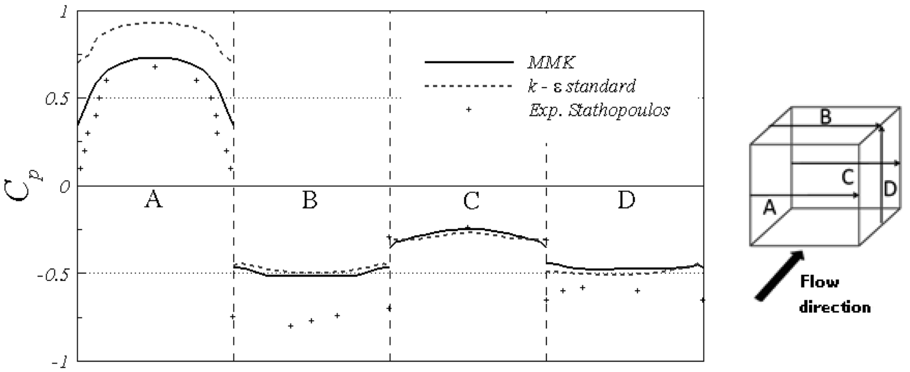

Fig. 9 Same as Fig. 8 for secondary alignments of the faces

difficulty in reproducing the inversion of the flow direction inside the separation bubble near the suction faces.

The pressure coefficient distribution obtained for the ABL profiles adopted by the EC1 is represented in Figs. 10(a) to 10(d). With a uniform incident flow the stagnation point is adjacent to the ground (see Fig. 7), whereas in the terrain type zero profile it is located at $^2/_3$ of the windward face height. The stagnation point is located increasingly higher for the profiles correspondent to more rough terrains, being located at about $^3/_4$ for the terrain type III profile. It was not possible to obtain a realist pressure distribution correspondent the type IV terrain, due to the significant change of this profile between the inlet boundary and the region near the obstacle, as seen in Fig. 2.

4. Conclusions

In this paper, the numerical calculation of the wind action on buildings is addressed, focusing on the influence of the use of the mean velocity and turbulence profiles in the calculation. This study aims to show the difficulties and strategies needed to apply the ABL-type profiles for the calculation of the flow around buildings, using CFD commercial codes.

In the first set of results obtained in this work, where only the part of domain upstream the obstacle was used, it was verified that the usage of the correct boundary condition on the floor is essential to obtain a stable profile. However, the usage of numerical roughness available in FLUENT has revealed to be insufficient to maintain the velocity profile, as the code is not prepared for the representation of large-scale aerodynamic roughness.

To determine the pressure coefficients on the cubic building's walls, it was necessary to change the definitions of the standard $k-\varepsilon$ model applying the MMK alteration. This was needed to prevent the high turbulence in the impinging region, that originated coefficients higher than 1 in the windward face. However, it was noticed in the runs with an empty domain that this modification has increased the dissipation of the ABL-characteristic turbulence along the domain. With this modification, values closer to the experimental ones were obtained, although the flow inside the suction bubble near the sharp eaves was still not accurately represented.

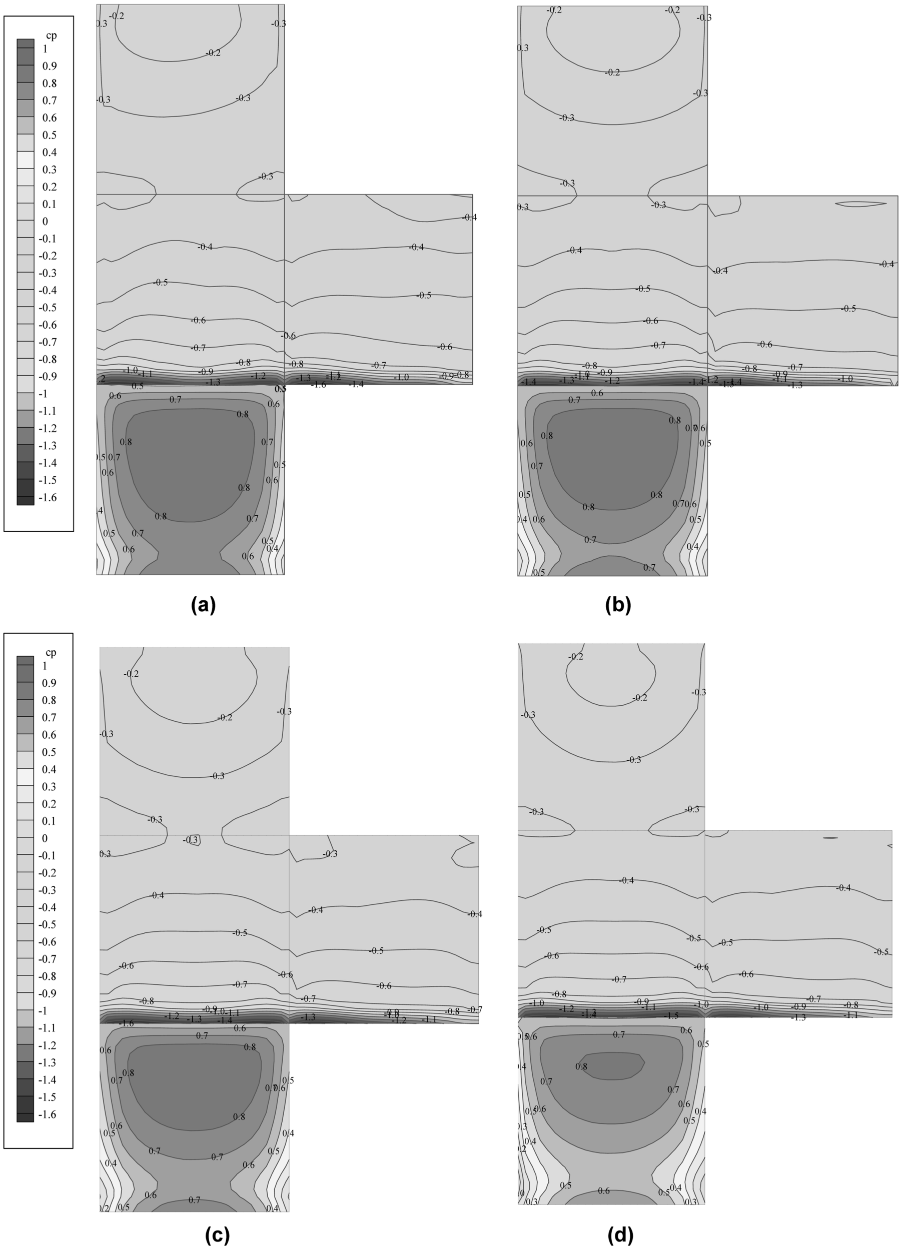

Fig. 10 C_p values on the faces of the cube for incident ABL profiles flow with the MMK modification of the turbulence model. (a) Terrain Type 0, (b) Terrain Type I, (c) Terrain Type II and (d) Terrain Type III

References

Blocken, B., Stathopoulos, T. and Carmeliet, J. (2007), "CFD simulation of the atmospheric boundary layer: wall function problems", *Atmos. Environ.*, **41**, 238-252.

Castro, I.P. and Robbins, A.G. (1977), "The flow around a surface-mounted cube in uniform and turbulent streams", *J. Fluid Mech.*, **79**, 307-335.

Eurocode 1 (2004), *Actions on structures - General Actions - Part 1-4*, Wind Actions.

Fluent Inc. (2005), *FLUENT 6.2 User's Guide*, USA.

Hargreaves, D.M. and Wright, N.G. (2007), "On the use of the κ-ε model in commercial CFD software to model the neutral atmospheric boundary layer", *J. Wind Eng. Ind. Aerod.*, **95**, 355-367.

Huang, S., Li, Q.S. and Xu, S. (2007), "Numerical evaluation of wind effects on a tall steel building by CFD", *J. Constr. Steel Res.*, **63**, 612-627.

Li, Q.S., Zhi, L.H. and Hu, F. (2009), "Field monitoring of boundary layer wind characteristics in urban area", *Wind Struct.*, **12**, 553-574.

Murakami, S., Mochida, A., Hayashi, Y. and Sakamoto, S. (1992) "Numerical study on velocity-pressure field and wind forces for bluff bodies by κ-ε, ASM and LES", *J. Wind Eng. Ind. Aerod.*, **41-44**, 2841-2852.

Richards, P.J. and Hoxey, R. (1993), "Appropriate boundary conditions for computational wind engineering models using the κ-ε model", *J. Wind Eng. Ind. Aerod.*, **46-47**, 145-153.

Stathopoulos, T. and Dumitrescu-Brulotte, M. (1985), "Design recommendations for wind loading on structures of intermediate height", *Can. J. Civil Eng.*, **16**, 910-916.

Stathopoulos, T. (1999), "The numerical wind tunnel for industrial aerodynamics: real or virtual for the new millennium?", *J. Wind Eng. Ind. Aerod.*, **81**, 347-360.

Tsuchiya, M., Murakami, S., Mochida, A., Kondo, K. and Ishida, Y. (1998), "Development of new k-ε model for flow and pressure fields around bluff body", *J. Wind Eng. Ind. Aerod.*, **67-68**, 169-182.

CC