MI/PMC 2012
2012 Joint Conference of the Engineering Mechanics Institute and the
11th ASCE Joint Specialty Conference on Probabilistic Mechanics and Structural Reliability
June 17-20 2012, Notre Dame, IN

ASSESSING THE EFFECT OF ROUGHNESS BOUNDARY CONDITIONS ON SIMULATING ATMOSPHERIC BOUNDARY FLOW

Daniel Abdi[1], Girma T. Bitsuamlak[2]
[1] PhD Student, Dept. of Civil and Environ. Eng.; Florida Intl. University, 10555 W. Flagler Street, Miami, FL 33174. Ph: 786-217-4276, Fax: 305-348-2802, Email: dabdi001@fiu.edu
[2] Assistant Prof., Dept. of Civil and Environ. Eng.; Director, Lab. For Wind Eng. Research, Intl. Hurricane Research Center; Florida Intl. University, 10555 W. Flagler Street, Miami, FL 33174. Ph: 305-733-1993, Fax: 305-348-2802, Email: bitsuamg@fiu.edu

Abstract

Simulation of atmospheric boundary layer (ABL) flow in computational wind engineering (CWE) requires accurate specification of velocity and turbulence intensity profiles at inlets compatible with the turbulence model, a compatible wall function at ground to sustain horizontal homogeneity of flow, and a driving shear stress at the top to compensate for decay of energy. In practice some of these requirements are often neglected and/or inconsistently specified resulting in incorrect simulation results. A wall function incompatible with the roughness characteristic used to generate the inlet profiles will result in streamwise gradients. The challenges in simulating ABL flow are reviewed and recommendations will be made. Use of wall functions for rough surfaces puts a requirement on the size of the first cell $Y_p > K_s$. This results in large cell sizes which conflicts with the requirement that a fine mesh be used close to the wall. This work will investigate an alternative approach of direct simulation with roughness blocks to incorporate the effect of roughness as suggested by Blocken et.al (2006). A low value of K_s can be used to model the surface of the block keeping the size of the first cell to a minimum. Also Wang et.al (2007) has investigated the development of internal boundary layer from multiple upstream roughness patches of different characteristics. Direct flow simulations with roughness blocks of different height, spacing and fetch length are carried out to investigate this phenomenon.

1 Introduction

Boundary conditions are applied at cutoff planes that divide the area we are interested in simulating from that we do not want to include in the simulation. However the average influence of the surrounding should be included in the model for realistic results. The type of boundary condition also affects the placement of the cutoff planes relative to the central region where obstacles are placed. For example, it is well known that use of symmetry boundary condition channels the flow thereby introducing artificial accelerations at the top of the building. The computational domain is divided into three regions (Blocken et al 2006) namely the central region where the obstacle is modeled as best as possible, and the upstream and downstream regions where the effect of obstacles is approximated through roughness elements. In wind tunnel testing, the latter regions are modeled using roughness blocks. However in CWE explicit modeling of roughness blocks is not common due to associated computational cost. This work will investigate this least explored but somewhat straight forward option of direct simulation to account for roughness. Wall functions, derived from the law of the wall, are commonly used to approximate the roughness effect of the wall on the wind flow. Roughness is specified either through aerodynamic roughness length z0 or equivalent sand grain roughness height Ks. Different rough wall functions for simulating ABL flow have been proposed in literature (Nikurdase 1933, Cebeci and Bradshaw 1977).

At the inlet of the computational domain fully developed equilibrium velocity and turbulence intensity profiles are usually applied. The profiles used should be consistent with the upstream surface roughness characteristics (Davenport 1961, Wieringa 1992). The velocity specified at the inlet should be maintained within the computational domain until it reaches the face of the building. A peculiar problem in ABL simulations is that maintaining horizontal homogeneity is very difficult to achieve with current breed of CFD software. Richards and Hoxey 1993 have investigated this problem thoroughly and came up with boundary conditions for the inlet that satisfy horizontal homogeneity. Their result has been used by the wind engineering community for many years. However, it is not enough to specify inlet conditions to get a streamwise homogeneous flow. The wall functions used at the surface should be compatible with the roughness of the upstream fetch outside the domain. Failing to do so will result in development of internal boundary layer that starts from the inlet plane.

At the sides and top of the domain, a symmetry boundary condition that prevents inflow or outflow is usually applied. This boundary conditions results in a parallel flow at the boundary which could sometimes lead to artificial acceleration if enough space is not provided between the obstacles and the boundary plane. To solve this problem the domain is sized in such a way that blockage ratio is set at a certain limit below which the effect is minimal. Another solution is to replace the boundary condition with one that allows flow outwards through the boundary (Franke et.al 2004).

The common use of symmetry boundary condition at the top of the boundary is rather unfortunate since it ignores the contribution of geostrophic wind in driving the ABL flow. Many researchers have noted that use of symmetry boundary condition results in streamwise gradients of velocity profile. However there are many reasons why symmetry is assumed in many wind engineering problems (O'Sullivan et.al 2011). The major physical reason is that log layer in the ABL extends only up to a certain depth above which the gradient of velocity becomes zero. Also it is not known a priori what the values would be set at the top if symmetry boundary condition is not used. A shear stress boundary condition ($\tau = \rho u^{*2}$) should be applied at the top to get a homogeneous (non-decaying) profile (Richard and Hoxey 1993, Hargreaves 2006). Another approach used by Blocken et.al 2007 is to apply dirichlet boundary condition for velocity and turbulence quantities at the top.

2 Simulation on an empty fetch

To demonstrate the problem with maintaining an ABL velocity profile introduced at the inlet throughout the length of the domain, the example provided in Hargreaves et.al (2007) is redone using CFD software developed by the authors. Hargreeves et al have noted that use of commercial software with default wall boundary conditions will likely suffer from the problem of inhomogeneous profiles. The computational domain is shown in Figure 1. The 5km fetch length allows any inlet effects to dissipate before reaching the outlet. A mean wind speed of 10m/s at reference height of 6m is assumed. The surface roughness length z_0 is 0.01, from which the sand grain roughness K_s is calculated.

The k-epsilon turbulence model is used with inlet boundary conditions as specified by Richard and Hoxey. A compatible wall function should be applied at the bottom to help in development of homogeneous profile along the fetch length. The formulae used by Richards and Hoxey for the k-epsilon model is shown below in equations (1) – (3).

$$u = \frac{u_*}{\kappa} \ln\left(\frac{z + z_0}{z_0}\right) \tag{1}$$

$$\kappa = \frac{u_*^2}{\sqrt{C_\mu}} \tag{2}$$

$$\varepsilon = \frac{u_*^2}{\kappa(z + z_0)} \tag{3}$$

The above three equations satisfy the standard k-epsilon equation for a von-karman constant of 0.4. The standard wall function of Launder and Spalding (1974) can only be applied for smooth walls. If the roughness elements are to be simulated using array of blocks, as will be done in the later part of this work, then assuming a smooth surface is acceptable. However for the current simulation of empty fetch, the roughness is incorporated through modification of the wall function itself.

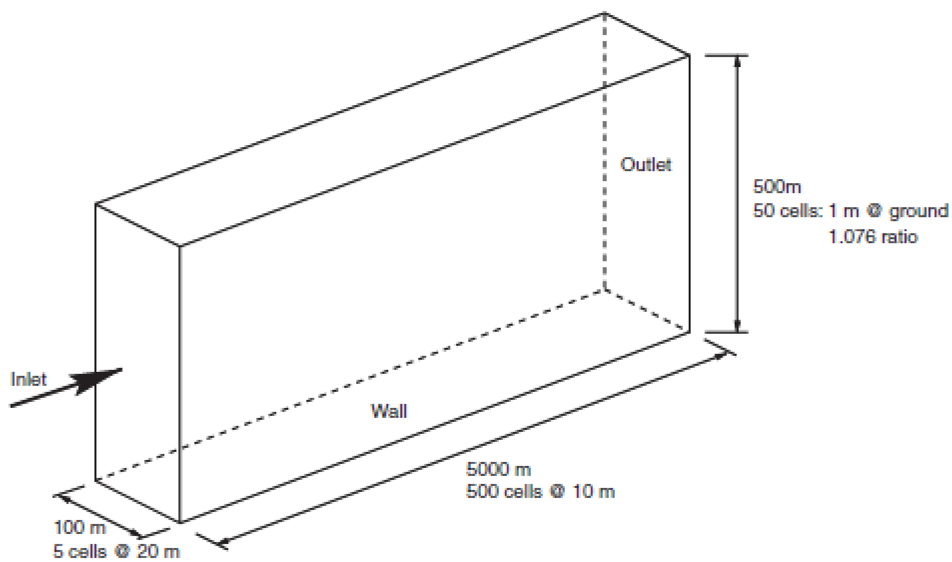


Figure 1. Hargreaves et al. computational domain for an empty fetch

The log-law can be modified for rough wall surfaces by just adding an extra term on the right hand side ΔB which is a function of sand grain roughness K_s. Nikurdase (1933) conducted extensive experiments on rough wall surfaces and found out that the log-law has still the same slope when plotted on semi-log scale i.e. 1/κ.

$$u^+ = \frac{1}{\kappa}\ln(Ey^+) - \Delta B \tag{4}$$

For fully rough flow with (K_s^+ >= 90), the following approximation for ΔB is suggested by Cebeci and Bradshaw. K_s^+ is dimensionless sand grain roughness (K_s u* / ν).

$$\Delta B = \frac{1}{\kappa}\ln(1 + C_{ks}K_s) \tag{5}$$

At the sides and top symmetry boundary condition is applied. At the outlet the pressure is fixed to 1 atm and the velocity gradient is set to 0.

2.1 Incompatible wall roughness simulation

The first simulation applies the Richard and Hoxey boundary conditions but assumes a smooth surface thereby creating a situation where the surface roughness exhibits a sudden jump at the inlet. Due to this incompatibility, stream wise gradients are observed in the profiles of U, k and epsilon as shown in fig 2.

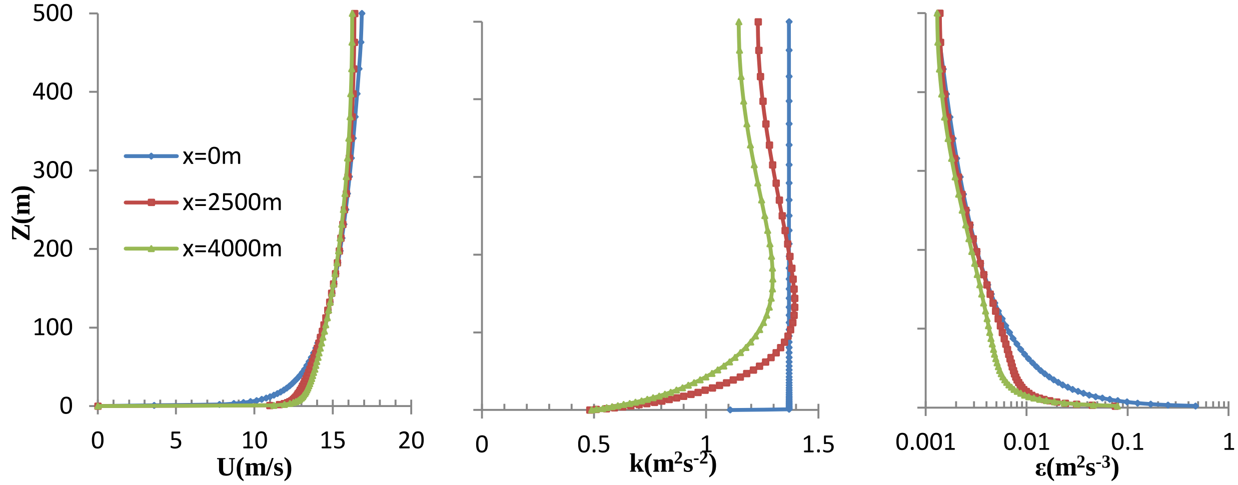


Figure 2. Plots of velocity, turbulent kinetic energy and turbulent dissipation for case 2.1

Close to the ground, both the velocity profile and turbulence dissipation show large changes as one goes downstream; while the profiles towards the top remain somewhat constant. On the other hand, the turbulent kinetic energy shows variations throughout. The difficult of maintaining the turbulent kinetic energy along the fetch has been noted by Richards et.al.(2011).

2.2 Compatible wall roughness simulation

When a wall roughness compatible with the inlet velocity profile is applied, both the velocity profile and turbulence intensity profile are maintained throughout the fetch as shown in Figure 3. The sand grain roughness used for the simulation is determined according to the relation $K_s = 20z_0 = 0.2$ and $C_{ks} = 0.5$. The calculated turbulent kinetic energy profile still shows variations from a vertical line.

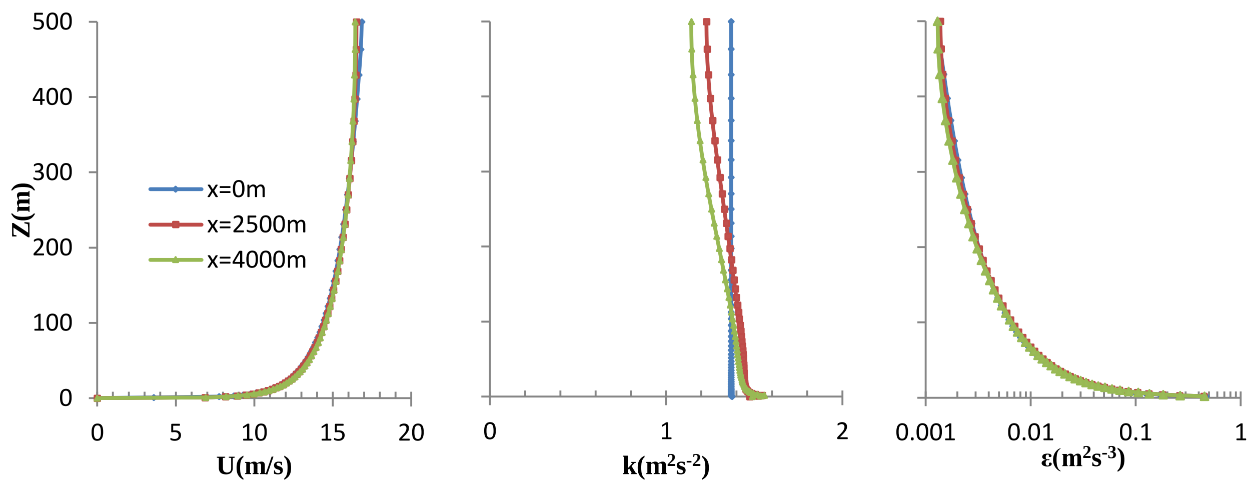


Figure 3. Plots of velocity, turbulent kinetic energy and turbulent dissipation for case 2.2

2.3 Dirichlet boundary condition at the top

From the previous simulations, we observe that the flow quantities at the top show some variations due to the imposed symmetry boundary condition. Blocken et.al (2006) has suggested using dirichlet boundary condition to make sure that the flow quantities remain the same at least at the top of the boundary. The result for this case is shown in Figure 4. While velocity and turbulence dissipation show an almost perfect fit from start to finish of the fetch, the turbulent kinetic energy profile show a rather distorted profile compared to the previous cases. Other simulations have been carried out which confirm the same observation.

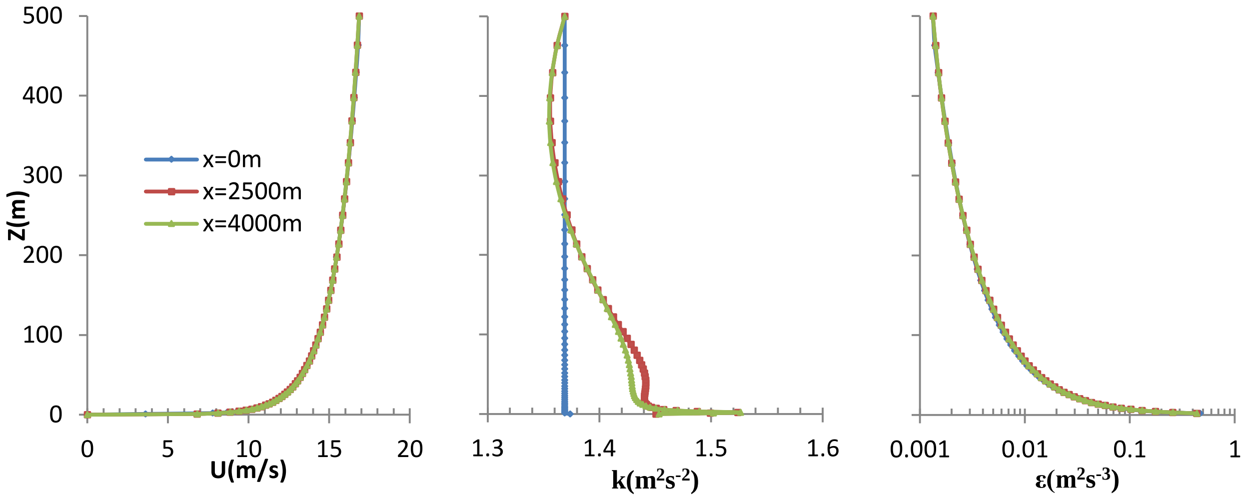


Figure 4. Plots of velocity, turbulent kinetic energy and turbulent dissipation for case 2.3

2.4 Dirichlet boundary condition for k and epsilon at the inlet

It is customary to specify the values of k and epsilon to be fixed to a constant at the inlet. The assumption is correct for k but not for epsilon. The simulation result for this case shows a developing epsilon profile along the fetch, before reaching more or less the same values at the outlet.

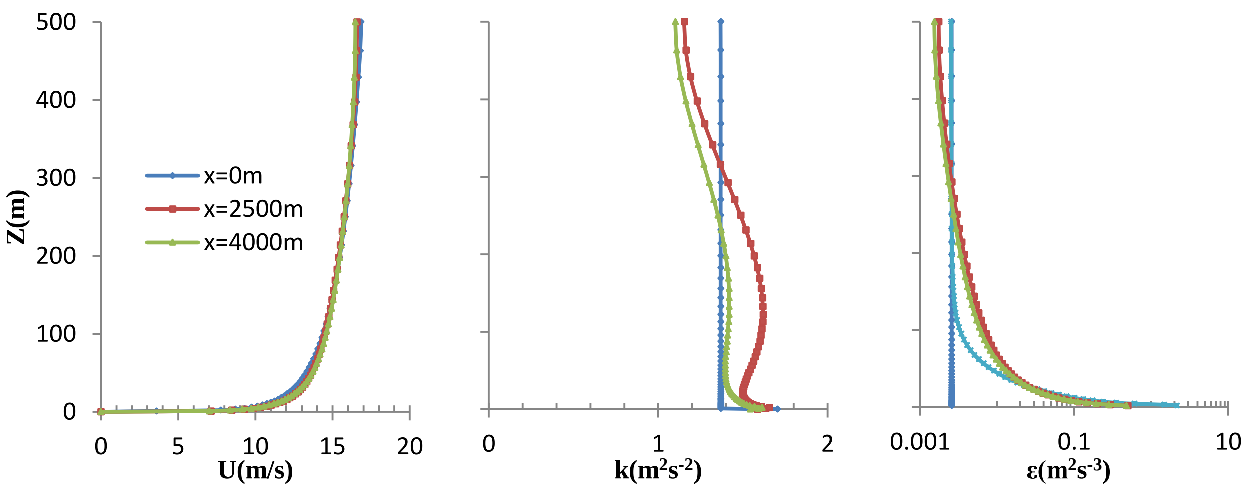


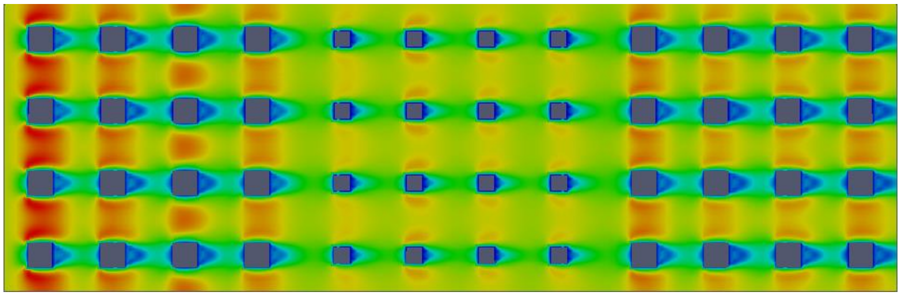
Figure 5. Plots of velocity, turbulent kinetic energy and turbulent dissipation for case 2.4

So far we have managed to get homogeneous velocity and epsilon profiles. To get a homogeneous turbulent kinetic energy profile further modifications to the wall function is necessary as described in Hargreaves (2007). Most commercial code do not generally offer wall functions that will maintain the k profile, however it is possible to implement the modifications to get a homogeneous profile for k.

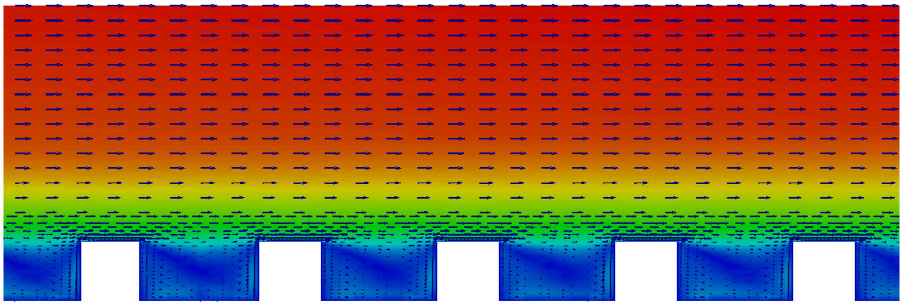
3 Explicit modeling of roughness elements

Use of wall functions for rough surfaces puts a requirement on the size of the first cell $Y_p > K_s$. This usually results in large cell sizes which conflicts with the requirement that a fine mesh be used close to the wall. The second part of the study investigates an alternative approach of direct simulation with roughness blocks to incorporate the effect of roughness as suggested by Blocken et.al 2006. This is in line with the common use of roughness elements (blocks) in experimental boundary layer wind tunnel (BLWT). A low value of K_s can be used to model the surface roughness of the block while keeping the size of the first cell to a minimum. A drawback of this method is increased computational cost due to explicit modeling of roughness elements. In some cases, it could be practically impossible to get results within a reasonable time when 3D models and complex turbulence models are used.

For a given ABL profile, simulations are carried out for different configuration of roughness elements until the desired profile is obtained at a target location downstream. This iterative process could be time consuming if previous data is not available from wind tunnel or CFD studies. If a regular array of blocks arranged in simple manner (aligned or staggered) is used, the inherent symmetry can be exploited to reduce the computational cost significantly. A typical symmetrical simulation over a multi-patch roughness is shown in Figure 6 below.



(a) Plan of 4 x 16 roughness blocks



(b) Profile along mid section of roughnes blocks

Figure 6. Mean velocity profile for a three patch roughness elements (k-epsilon model).

It is clear that simulating one raw of obstacle arrays will suffice for a time averaged turbulence model such as the k-epsilon model. A section passing through the centre of the cubes and another one passing through the centre of the open space between two raws will yield the same result when the number of raws is sufficiently large. A two dimentional simulation can sometimes be used, but in most cases it results in larger spacing of roughness blocks for the common roughness categories shown in Table 1.

Table 1. Roughness length and power law coefficients

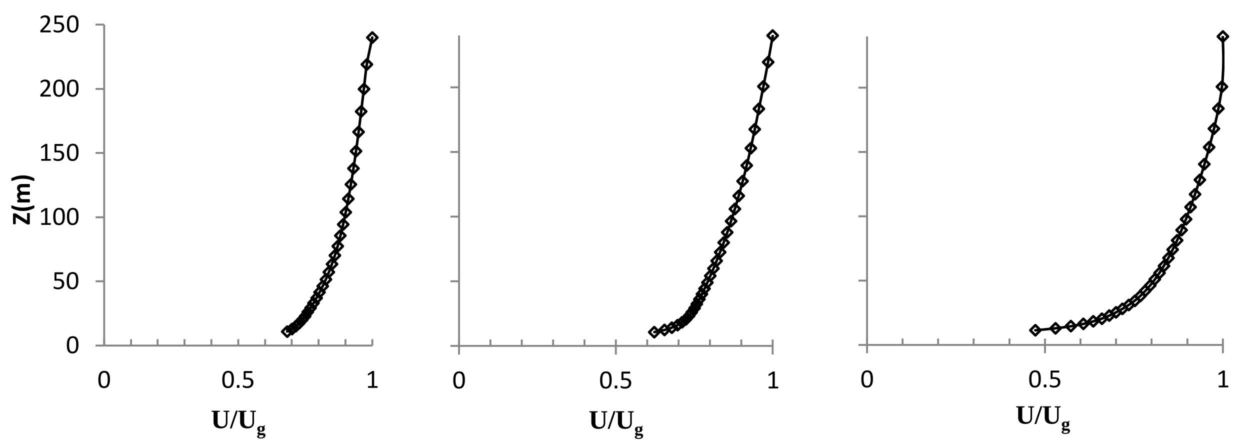
Category	Z_0	α	I_u(10m)
Open Country	0.024	0.14	0.17
Sub urban	0.42	0.26	0.28
Urban	1.03	0.32	0.35

An approximate forumula to relate roughness length with average frontal and planar area of obtacles can be found in MacDonald et al (1996) eq (6)-(7). The spacing and height of blocks using the formula are usually good estimates for starting the iterative process. The first simulations conducted are for homogeneous roughness patches of open-country, sub-urban and urban roughness characteristics. Steady state simulations with k-epsilon turbulence model are conducted on patch length of about 1km for each of the roughness patches using 18 regularly arranged cubic obstacles. The result of this preliminary analysis ,velocity profile and turbulence intensity profiles, for the three categories of roughness are shown in Figure 7.

$$\frac{z_0}{H} = \exp(-0.52\lambda_f^{-0.5}) \quad (6)$$

$$\lambda_f = \frac{A_f}{A_d} = \frac{1}{(1+S/H)^2} \quad (7)$$

Once the configurations for the basic roughness categories are determined, multiple roughness patches of different characteristics can be simulated. Wang et al (2007) has done extensive work in the wind tunnel to study the effect of inhomogeneous roughness on wind loads of buildings. However, the numerical study they conducted did not use explicitly modeling of roughness elements; rather a shear stress is directly applied at the bottom. In present study we have tried to simulate most of the 67 cases done by Wang using the explicit modeling approach. The run time of the simulation for running all the cases took about one month on a cluster of computers. Due to this large computational requirement, LES simulation cannot be carried out even though it is known to give better results for turbulence intensity calculations.



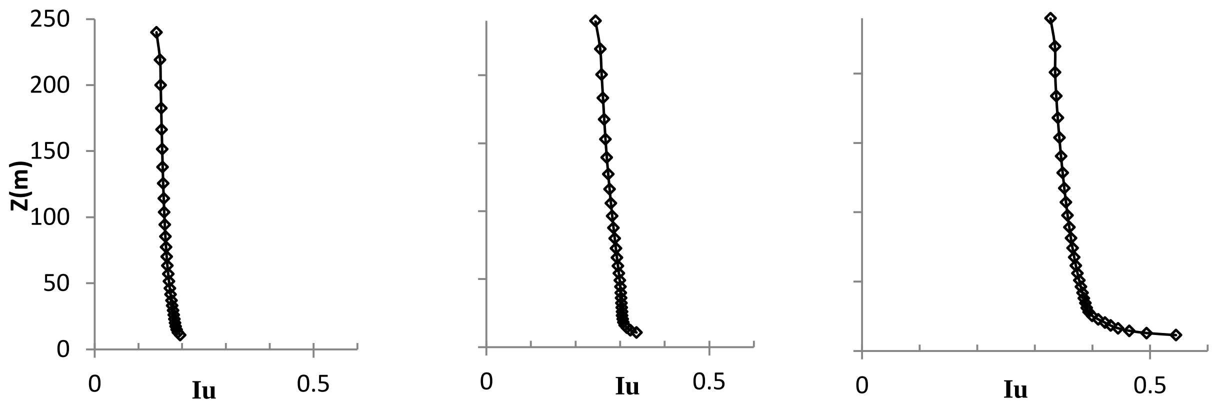


Figure 7. Normalized wind speed and turbulence intensity profiles for open-country (left), sub-urban (middle), and urban (right) roughness.

An example multi-patch simulation result with three patch roughness of urban ,suburban and urban roughness of each 1km long is shown in Figure 8. The profile measurements are done at three sections : mid-section of blocks (y = 50m), mid-section of open-space between blocks (y = 0m) and at y = 25m. The velocity profile over the blocks will be displaced by approximately the height of the blocks at the first section. It is therefore necessary to take the average of profiles through the three sections when comparing results with other wind speed models such as Wang and ESDU models. With this procedure most of the cases studied by Wang 2007 has been simulated, and comparison with the previous two wind speed models is in progress.

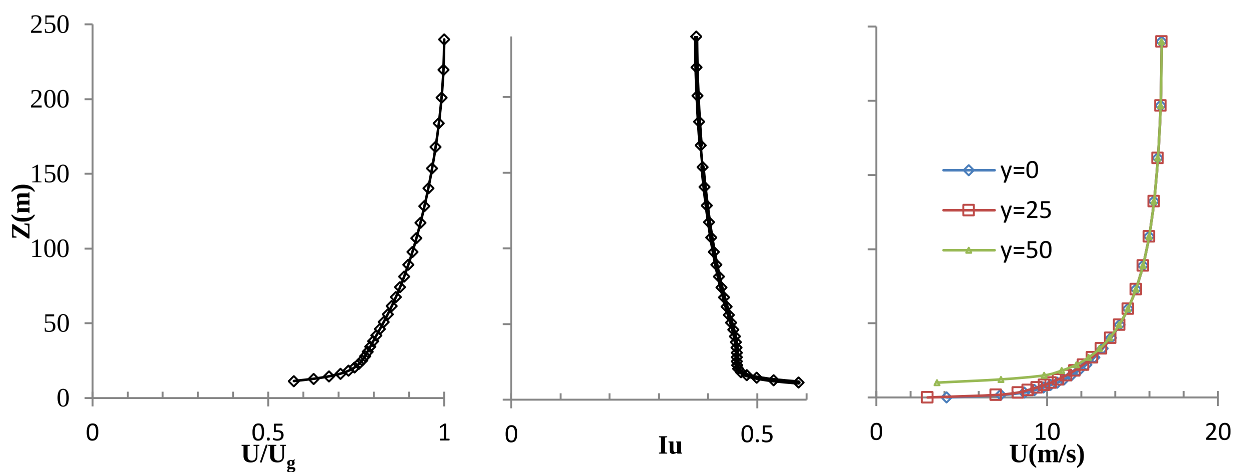


Figure 8. Average normalized wind speed, average turbulence intensity profile and wind speed at three sections of a three patch Urban-Suburban-Urban roughness.

4 Conclusions

It has been shown that simulation of atmospheric boundary layer requires accurate specification of boundary conditons. Even in the simple case of flow over an empty fetch , sustaining a horizontally homogeneous profile is difficult. Simulations have been carried out to demonstrate this problem by making incremental changes to the boundary condition at the inlet and wall functions that lead to horizontal homogeneouty of wind profiles. However for moderately rough surfaces the method of using wall functions to impose surface roughness can not be used due to the requirement that larger cell size be used close to the wall. Hence explicit modeling of roughness blocks as suggested by Blocken et al. (2007)

is used in this study. First preliminary simulations are carried out to find the proper configuration that will produce profiles for typical open country, sub urban and urban patches. After that simulations on multiple patches of different roughness characteristics and length are carried out, similar to that conducted by Wang et al. (2007) in a wind tunnel experiment. The explicit modeling of roughness elements, while more expensive than using rough wall functions, can give better results at a much smaller cost than BLWT experiments.

References

Blocken, B., Stathopoulous,T.,Carmeliet, J. (2006), “CFD simulation of atmospheric boundary layer : wall function problems.” Atmospheric Environment 41 (2), 238–252.

Blocken, B., Carmeliet, J., Stathopoulos, T. (2006), “CFD evaluation of the wind speed conditions in passages between buildings—effect of wall-function roughness modifications on the atmospheric boundary layer flow.”,Journal of Wind Engineering and Industrial Aerodynamics.

Cebeci, T., Bradshaw, P. (1977), “Momentum Transfer in Boundary Layers”, Hemisphere Publishing Corporation, New York.

Davenport, A.G. (1961), “The application of statistical concepts to the wind loading of structures.” In: Proceedings of the Institution of Civil Engineers, August.

Franke, J., Hirsch, C., Jensen, A.G., Krus, H.W., Schatzmann, M., Westbury, P.S., Miles, S.D., Wisse, J.A.,Wright, N.G. (2004), “Recommendations on the use of CFD in wind engineering.”, In: van Beeck, J.P.A.J. (Ed.),Proceedings of the International Conference Urban Wind Engineering and Building Aerodynamics, von Karman Institute.

Hargreeves,D.M.,Wright,N.G. (2006) ,”On the use of k-epsilon model in commercial CFD software to model the neutral atomospheric boundary layer.”, Journal of Wind Engineering and Industrial Aerodynamics.

Launder, B.E., Spalding, D.B. (1974), “The numerical computation of turbulent flows”, Comput. Methods Appl. Mech. Eng. 3, 269–289.

Nikuradse, J. (1933), “Stro¨ mungsgesetze in rauhen Rohren”, Forschung Arb. Ing.-Wes. No. 361.

Richards, P.J., Hoxey, R. (1993), “Appropriate boundary conditions for computational wind engineering models using the k–epsilon turbulence model.” Journal of Wind Engineering and Industrial Aerodynamics. 46, 47, 145–153.

Richards, P.J., Norris,S.E. (2011), “Appropriate boundary conditions for computational wind engineering models revisted.”, Journal of Wind Engineering and Industrial Aerodynamics.

O’Sullivan,J.P.,Archer,R.A., Flay,R.G.J. (2010), “Consistent boundary conditions for flows within the atmospheric boundary layer. “, Journal of Wind Engineering and Industrial Aerodynamics.

Wieringa, J. (1992), “Updating the Davenport roughness classification”, Journal of Wind Engineering and Industrial Aerodynamics 41–44, 357–368.

Wang,K., Stathopoulos, T. (2007), “Exposure model for wind loading of buildings.”, Journal of Wind Engineering and Industrial Aerodynamics