10th ANKARA INTERNATIONAL AEROSPACE CONFERENCE **AIAC-2019-151**
18-20 September 2019 - METU, Ankara TURKEY

A NUMERICAL ASSESSMENT OF ATMOSPHERIC BOUNDARY LAYER SIMULATION INSIDE TWO DIFFERENT BOUNDARY LAYER WIND TUNNELS

Anas Abdulrahim[1]
Middle East Technical University
Ankara, Turkey

Munir Elfarra[2]
Ankara Yıldırım Beyazıt University
Ankara, Turkey

Oğuz Uzol[3]
Middle East Technical University
Ankara, Turkey

ABSTRACT

A new large scale wind tunnel is under development at METU Center for Wind Energy (RÜZGEM). This wind tunnel is a closed-loop multi-purpose wind tunnel with a 3 m x 7 m x 20 m boundary layer test section. Inside this test section the atmospheric boundary layer (ABL) will be simulated using the spire-roughness element technique in order to represent different terrain exposures (or categories) as defined by American Society of Civil Engineers (ASCE). Since no experimental data are available yet, Computational Fluid Dynamics (CFD) will be implemented as a tool in order to provide an initial assessment for the simulation of the ABL. However, in order to validate the CFD approach, another wind tunnel test case from literature will be used for comparison. This wind tunnel has 1.82 m x 1.82 m x 9.8 m test section. Four different test cases have been simulated and the results show reasonable agreement between the experiments and numerical results in terms of velocity profiles, power law exponents and boundary layer parameters.

INTRODUCTION

In order to correctly simulate the atmospheric boundary layer (ABL) inside a wind tunnel test section, certain conditions need to be met such as the velocity profile, turbulence intensity as well as turbulent length scales and power spectrum that represent a certain terrain category. According to ASCE7-10 (2010), there are four different terrain categories (A, B, C & D) classified according to the power law exponent (α) ranging from 0.4 for terrain A to 0.05 for terrain D.

Wind tunnel simulation of the ABL could be achieved using either passive or active techniques. Passive techniques include vortex generators such as spires, array of roughness elements, grids as well as a combination of these devices [Counihan, 1969 & 1973; Cook, 1973; Irwin, 1981]. On the other hand, active techniques include active grids, multiple fans, or oscillating spires [Cao et al., 2002; Pang and Lin, 2008]. Both techniques require intensive research and investigation in order to find the appropriate combination of these devices to properly simulate a certain terrain or exposure category. Therefore, in order to save time and cost, CFD has been implemented to provide initial assessment for such experiments. For instance, Shojaee et al., (2009 & 2014), conducted CFD simulations in the Ankara Wind Tunnel using different combinations of spires and roughness elements in order to represent different terrain

[1]PhD Candidate. Department of Aerospace Engineering, METU. Lecturer, Faculty of Air Transportation, University of Turkish Aeronautical Association (UTAA.) Email: anas.abdulrahim@metu.edu.tr
[2] Assistant Professor. Department of Aerospace Engineering, Ankara YBU. Email: monierelfarra@hotmail.com
[3] Professor. Department of Aerospace Engineering, METU. Head, METU Center for Wind Energy (RÜZGEM). Email: uzol@metu.edu.tr

exposures, and the results show good agreement with the experiments. Moreover, Abubaker et al. (2018) also conducted CFD analysis for two different wind tunnels with short test sections and the results were in agreement with the experiments. In addition, Yassen and Abdelhamed (2015), as well as Li et al., (2018) also conducted CFD simulations of ABL inside short test sections. These research activities show that CFD could be implemented as a design tool to simulate the ABL inside wind tunnel's test section especially during the initial design of the wind tunnels or if no experimental data are available.

The spire-roughness element technique is the most widely used method to simulate the ABL. Spires (sometimes also called vortex generators) are the main element for developing the boundary layer and their heights dictate the thickness of the boundary layer, whereas, roughness elements are used to improve the lower part of the boundary layer by adding more energy and momentum to the flow. Irwin, (1981), developed a design methodology for the spires and roughness elements based on theoretical and empirical data, which has been implemented in many wind tunnels and proved to be useful in providing an initial assessment of the ABL. However, the technique is not always successful in providing target boundary layer thickness or power law exponent as the theory suggests. Therefore, it requires a lot of fine tuning and modifications for the design and more tests in the wind tunnel in order to simulate a certain terrain exposure properly. As a consequence, CFD could be used instead for this initial assessment of the design in order to find the best combination of spires and roughness elements that will be used to simulate the ABL characteristics. Therefore, this paper will present initial assessment of the ABL to prove that CFD could be used for such studies with comparisons from the small wind tunnel experiments, as well as a sample test case for the RÜZGEM Large-Scale Wind Tunnel.

This paper presents the results of a CFD simulation study of the ABL, first inside a 1.82 m x 1.82 m x 9.8 m wind tunnel test section as given in Song, (2017). The CFD results were compared with selected measurement results from this wind tunnel. Similar CFD simulations are also performed for the 3 m x 7 m x 20 m boundary layer test section of the RÜZGEM Large Scale Wind Tunnel. Results include velocity profiles, contour plots and quantitative data summarizing the boundary layer characteristics such as displacement and momentum thicknesses as well as shape factors.

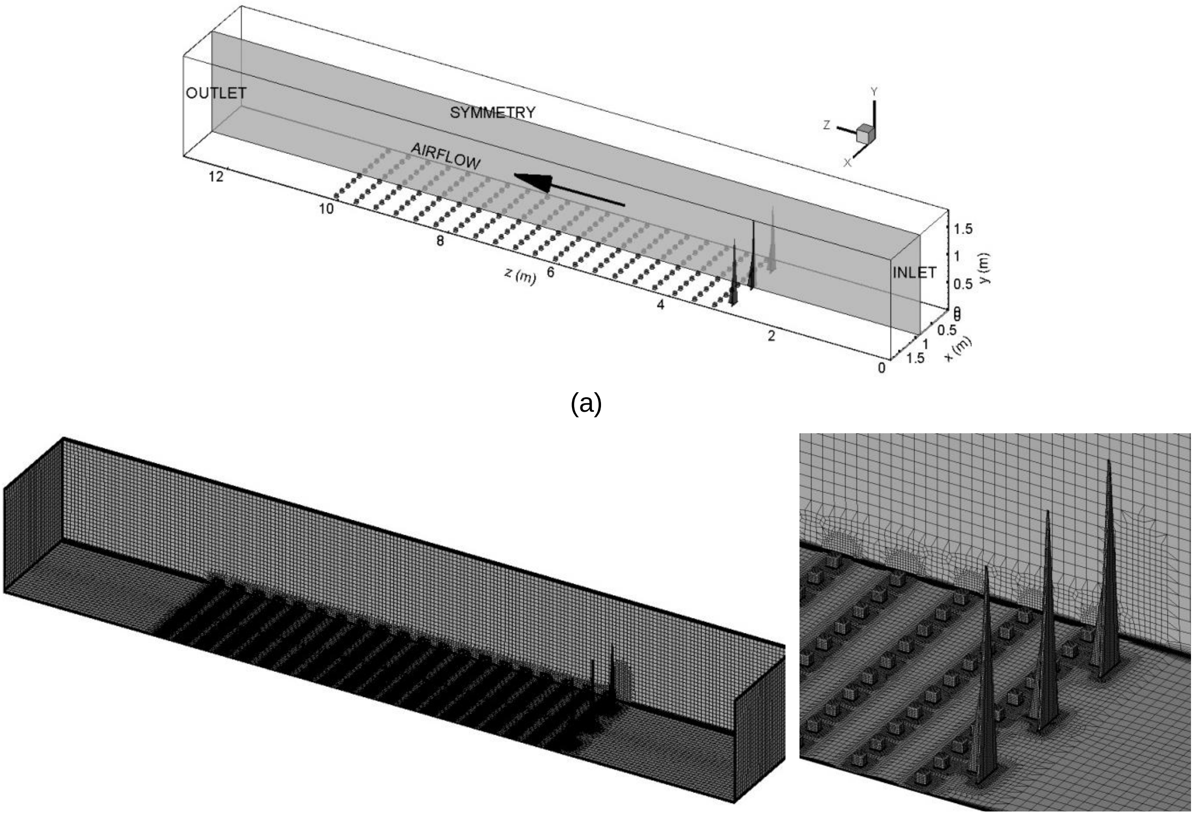
METHOD

Numerical Methodology

All CFD simulations in this study are conducted using the commercial CFD package FINE/Open developed by NUMECA International (2010). The FINE/Open solver is a 3-dimensional, unstructured, multi-block and multi-grid finite volume code. A sample computational domain and grid are shown in Figure 1. The grid is an unstructured hexahedral mesh with nearly 1 – 4.32 million cells depending on the test case considered. Half-models have been considered in order to reduce the domain size and computational time with symmetry imposed as shown in Figure 1a for the small wind tunnel.

3D steady-state compressible RANS equations with *k-ω M-SST* (Shear Stress Transport) turbulence model [Menter, 1994] have been solved with 2nd-order central numerical schemes and Merkle preconditioning to account for the very low speed condition [Choi and Merkle, 1993]. The working fluid (air) is treated as an ideal gas with viscosity obtained from the Sutherland law. Convergence criteria have been considered based on the outlet mass flow rate, where after sufficient number of computations the outlet mass flow rate remains steady.

Figure 1 shows the computational domain with the boundary conditions as well as the unstructured hexahedral mesh for spires with roughness element case with a closed-up view of the boundary layer resolved around the spires.



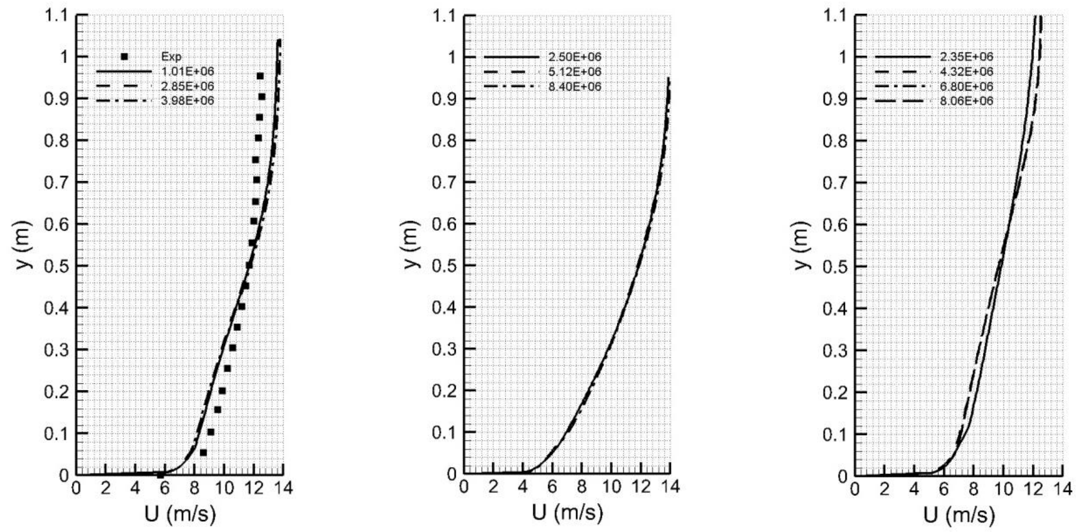
(a)

(b) (c)

Figure 1: (a) Computational domain, (b & c) unstructured hexahedral mesh with close-up view of the mesh around the spires and roughness element

Grid Independence Study

The results of a numerical simulation are generally dependent on the size of the mesh being used. A too coarse mesh will result in significant error, and as the mesh size gets finer this error should reduce as a consequence [Yassen and Abdelhamed, 2014]. However, if the size of the mesh elements is small enough that the numerical results are close to the experimental data, a further decrease in cell size should not affect the solution significantly. Therefore, in here we present a grid independence study in order to identify how coarse the grid can be without having significant errors. Figure 2 shows the velocity magnitude profiles for 3 refinement levels conducted for the 3-spires case, 3-spires with roughness element case, as well as for RÜZGEM Large Scale Wind Tunnel test case. Results show that there is no significant difference between the grid sizes, and this has proved that using a coarser mesh will suffice for the test cases considered.



(a) (b) (c)

Figure 2: Grid independence study: (a) 3-spires case, (b) 3-spires with roughness elements case, (c) RÜZGEM large-scale wind tunnel case

RESULTS AND DISCUSSIONS

Validation Study

The validation study is based on the study of Song (2017) where experiments are conducted inside a low-speed open return wind tunnel with cross-sectional area of 1.82 m x 1.82 m. The length of the boundary layer development section and the working test section is 9.8 m. Spires and roughness elements are located at the beginning of the development section. Different cases have been tested inside the wind tunnel for spires and roughness elements as well as combinations of both. It should be noted here that an additional 3 m was added upstream of the spires and roughness elements in order to compensate for the inlet contraction. Previous studies show that adding an extension will have no effect on the results as long as the distance between spires and measurement locations is kept the same [Amerio, 2014]. An additional reason is to allow the flow to initially develop before encountering the spires and cubes [Hobson-Dupont, 2015]. However, care should be taken not to make this section too long otherwise a boundary layer will develop upstream of the spires and roughness elements and this could affect the expected results.

The CFD simulations for the cases considered have been compared with the experimental results from Song, (2017). The velocity used for the test cases is 12 m/s as measured from the inlet of the boundary layer development section. Four test cases have been considered in the analysis (3 spires, 5 spires, roughness elements (20 x 10 array), 3 spires with roughness elements).

Figures 3 and 4 show the comparison of the velocity profiles obtained with CFD and the experimental results. The measurements were taken at the center of the turntable of the wind tunnel test section. As shown in the results there is a reasonable agreement between the CFD and experiments.

The velocity profiles show similar trends as well as comparable boundary layer thicknesses. Unlike the 5-spires case the 3-spire case seems to underpredict the velocity values in the lower part of the boundary layer, whereas, the roughness element case overpredicts the velocity values, this has been reflected in the quantitative results of the power law exponent, and boundary layer properties.

Table 1 shows a quantitative comparison of the CFD results with the experimental data using the boundary layer properties such as displacement thickness, momentum thickness and the shape factor. The boundary layer thickness is measured using the $0.99U_e$ (U_e: velocity at the edge of the boundary layer) rule. Moreover, using the velocity profiles the displacement thickness and momentum thickness have been calculated using the Trapezoidal rule for definite integrals.

Results show that the boundary layer properties are in general comparable in predictions with the experimental data, though there are differences. Nevertheless, one can conclude that CFD simulations could give a reasonable assessment of the effects of spires and roughness elements to be used in the RÜZGEM Large Scale Wind Tunnel boundary layer test section.

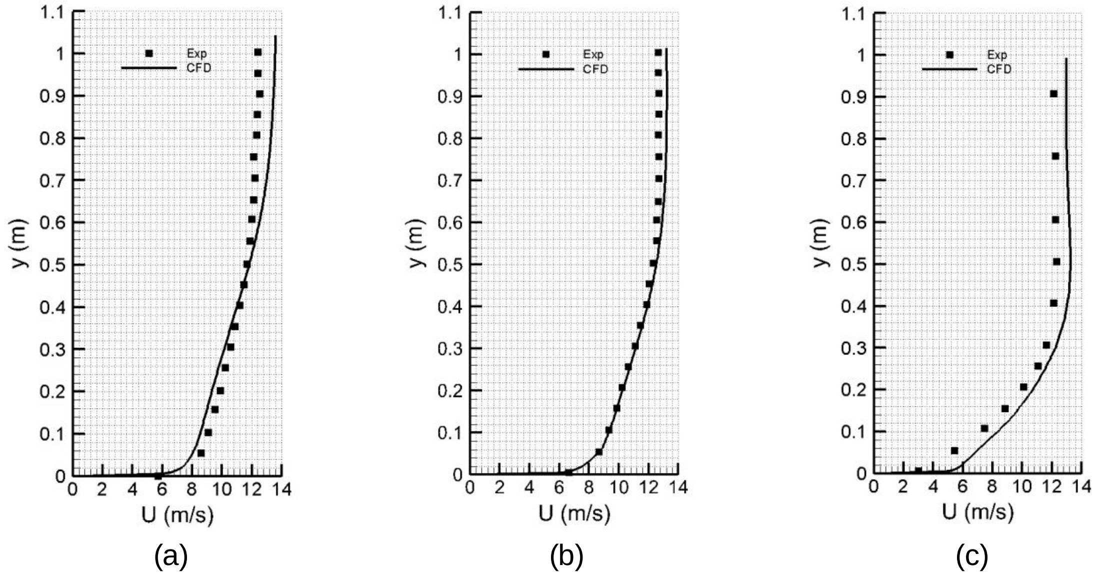


Figure 3: velocity magnitude profiles; (a) 3-spires case, (b) 5-spires case, (c) roughness elements (20 x10 array) case

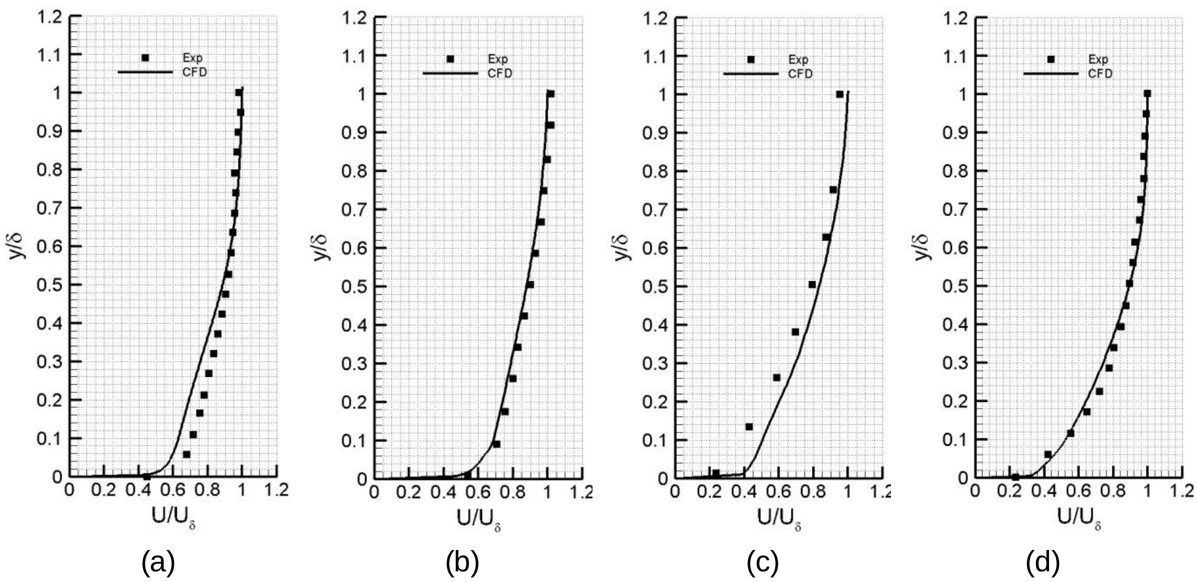


Figure 4: non-dimensional velocity profiles; (a) 3 spires case, (b) 5 spires case, (c) roughness elements (20 x10 array) case, (d) 3-spires with roughness elements

Table 1: Summary of the atmospheric boundary layer parameters

Test Cases	3-spires case		5-spires case		Roughness element case		3-spires + roughness elements	
Properties	Exp	CFD	Exp	CFD	Exp	CFD	Exp	CFD
Power Law Exponent (α)	0,15	0,19	0,14	0,17	0,36	0,28	0,27	0,29
Boundary Layer Thickness (δ) [m]	0,95	1,07	0,61	0,69	0,41	0,43	0,96	1,19
Displacement Thickness (δ^*) [m]	0,12	0,17	0,07	0,10	0,11	0,09	0,16	0,21
Momentum Thickness (θ) [m]	0,09	0,12	0,06	0,08	0,06	0,06	0,10	0,13
Shape Factor (H)	1,28	1,40	1,27	1,33	1,71	1,56	1,54	1,59

Figure 5 shows the velocity magnitude contours at the mid-plane section of the wind tunnel for the test cases considered. As expected the effect of spires is significant in the boundary layer and is felt all the way until the exit of the test section especially within the wake of the spires. Moreover, one can observe that as we move downstream and away from the spires the flow starts to recover to the freestream flow. The case of 3 spires shows significant effect than the 5 spires case since it has larger size. Similarly roughness elements also affect the boundary layer by adding more energy and momentum to the flow. For this reason, a combination of spires and roughness element could produce the desired boundary layer and power law exponent. As mentioned previously, the spires dictate the thickness of the ABL and the roughness element improves the lower part of the ABL in order to represent a certain terrain category.

Figure 6 also shows the velocity contours on the downstream plane of the spires and roughness elements. Mirror planes have been generated in order to show the complete flow field for better visualization. Results show that after a certain distance downstream of the spires the flow becomes developed and no significant changes occur in the velocity profiles and boundary layer thickness. In addition, the effect of the spires diffuses in the freestream flow as one would expect. However, the larger the spire height the more significant the effect will be on the flow field and the longer it takes for the spire effect to disappear.

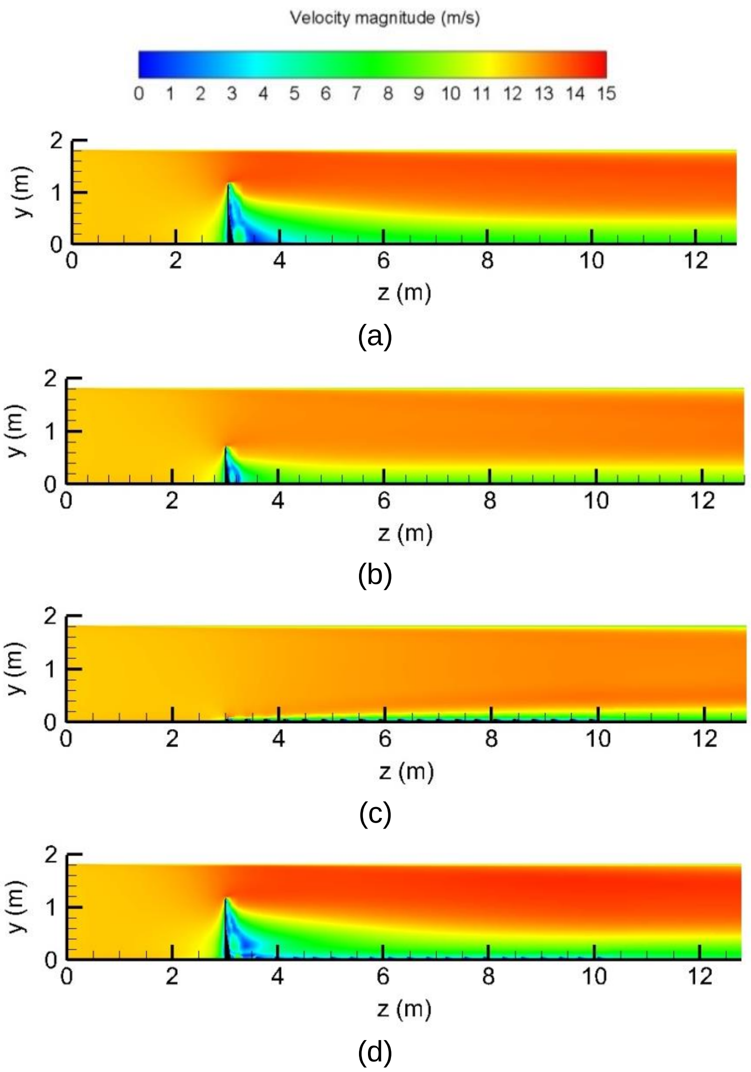


Figure 5: Velocity magnitude contours at the mid-plane: (a) 3-spires case, (b) 5-spires case, (c) Roughness elements (20x10 array) case, (d) 3-spires with roughness elements

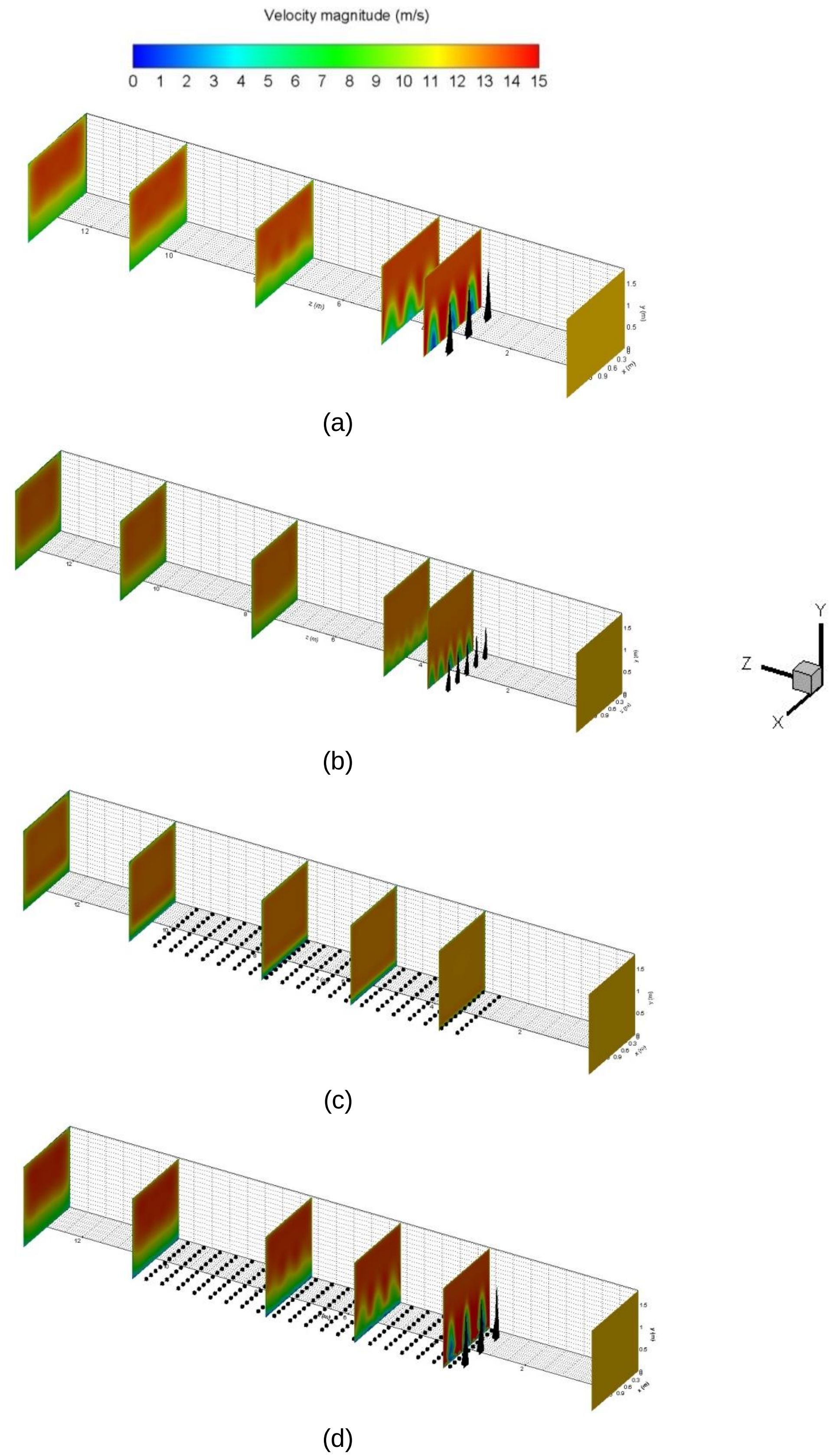


Figure 6: Velocity magnitude contours: (a) 3-spires case, (b) 5-spires case, (c) Roughness elements (20x10 array) case, (d) 3-spires with roughness elements

RÜZGEM Large-Scale Wind Tunnel Boundary Layer Test Section Simulations

The CFD simulations of the RÜZGEM Large Scale Wind Tunnel have been conducted using the same numerical approach of the validation study. However, due to the size of the wind tunnel the computational grid is expected to be large. Similarly half-model for the test section was implemented in order to reduce the number of elements in the grid by use of symmetry. The spires were specifically designed to produce a 1 m boundary layer thickness at 12 m/s inlet velocity. Figure 7 shows the computation grid with unstructured hexahedral mesh and a close-up view of the spires.

The spires were designed using Irwin's approach [2], which resulted in 12 spires for the test case considered. The aim of the approach is to produce a 1 m boundary layer thickness, based on a power law exponent of 0.4 (*terrain exposure A*). Since half-model was used in the numerical study, only 6 spires are shown in Figure 7. The spires were located 3 m downstream of the test section inlet. The measurement location is taken at the center of the turntable.

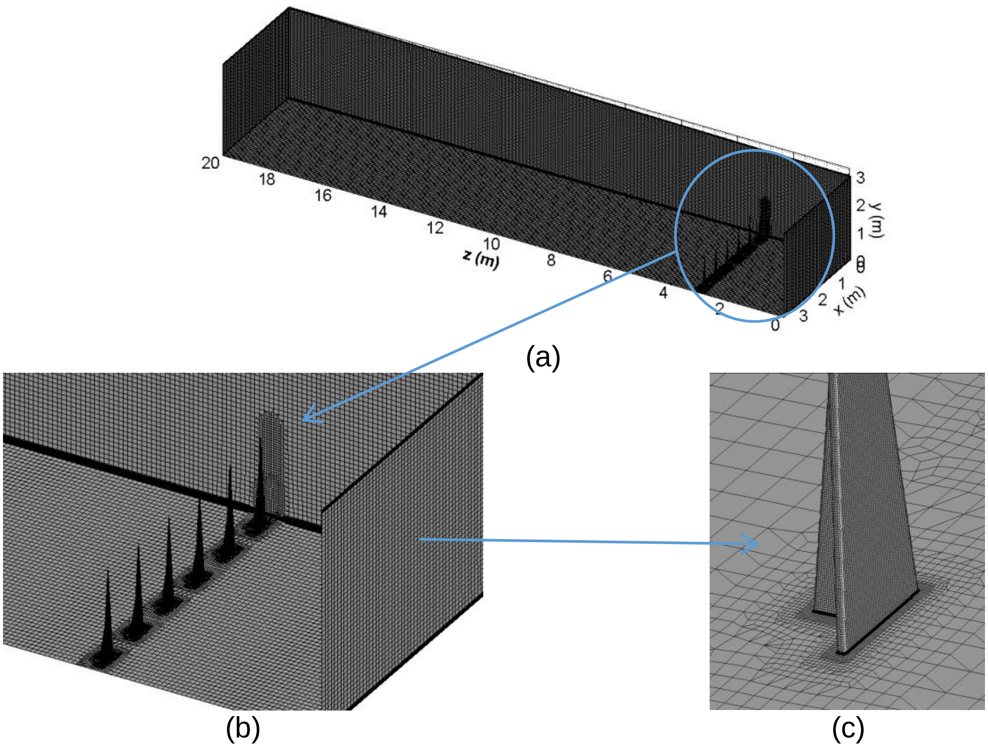
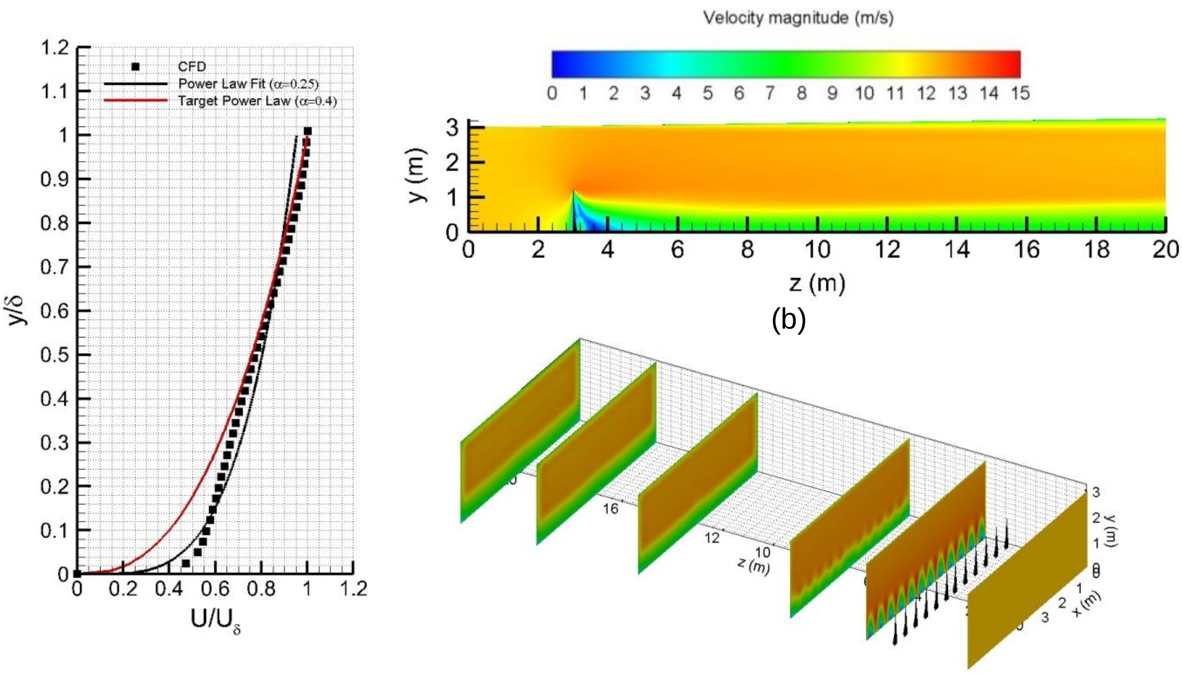


Figure 7: (a) unstructured hexahedral mesh, (b, c) close-up view of the spires and mesh around the spires

Figure 8 shows sample results of the simulations including the non-dimensional velocity profile at 13 m downstream of the spires as well as the velocity magnitude contours. A power-law curve is fit to the predicted velocity profile and compared to the power law velocity profile for terrain exposure category A (α=0.4). One can observe that although a boundary layer of 1 m thickness could be generated due to the presence of the spires, correct power law for terrain exposure A is not obtained. This is mainly due to the fact that Irwin's (1981) approach fails to capture exactly the terrain exposure using spires only, as expected. Previous studies show that Irwin's approach takes into account the use of roughness element in combination with spires, which could improve the result and achieve the target power law for terrain exposure A. The velocity contour plots also show flow field downstream of the spires, as expected the effect of spires after a certain distance disappears and the boundary layer is fully developed for the test case considered.



(a) (c)

Figure 8: (a) non-dimensional velocity profile, (b) velocity magnitude contours at mid-plane, (c) velocity magnitude contours downstream of the spires

Conclusions

In this paper we presented the CFD simulations of the ABL inside wind tunnel test section for two different wind tunnels. For the small wind tunnel which is used for validation, results show considerable agreement with the experiments for the cases considered, which proved that CFD could be implement for such studies. In addition, for RÜZGEM large-scale wind tunnel, the same CFD approach was implemented to attempt to generate a 1 m boundary layer thickness that represents terrain exposure A. Although a 1 m thick boundary layer was generated the target power law exponent failed to be captured. However, this study show that further investigation is necessary to try different combinations of spires and roughness elements in order to represent a certain terrain exposure properly.

References

Abubaker, A., Kostic, I., Kostic, O., (2018) *Numerical Modeling of Velocity Profile Parameters of the Atmospheric Boundary Layer Simulated in Wind Tunnels*, Materials Science and Engineering (393) conference series.

Abubaker, A., Kostic, I., Kostic, O., Stefanovic, Z., (2018) *CFD Modeling of Atmospheric Boundary Layer Simulations in Wind Tunnels*, Technicki Vjesnik (25) 6:1595-1602.

American Society of Civil Engineers (ASCE), (2010) *Minimum Design Loads for Buildings and Other Structures*, ASCE7-10.

Amerio, L., (2014) *Numerical and Experimental Analysis of Peak Pressure Loads on Rectangular Building*, PhD Thesis, Politecnico Di Milano.

Cao S., Nishi A., Kikugawa H., Matsuda Y., (2002) *Reproduction of Wind Velocity History in a Multiple Fan Wind Tunnel*. Journal of Wind Engineering and Industrial Aerodynamics 90:1719-1729.

Choi, D., Merkle, C.L., (1993) *The Application of Preconditioning in Viscous Flows*, Journal of Computational Physics vol.105:207-223.

Cook, N.J., (1973) *On Simulating the Lower Third of the Urban Adiabatic Boundary Layer in a Wind Tunnel,* Atmos. Environ 7:691-705.

Counihan, J., (1969) *An Improved Method of Simulating an Atmospheric Boundary Layer in a Wind Tunnel,* Atmos. Environ 3:197-214.

Counihan, J., (1973) *Simulation of an Adiabatic Urban Boundary Layer in a Wind Tunnel,* Atmos. Environ 7:673-689.

Hobson-Dupont, M., (2015) *The Development of a Small Scale Wind Tunnel Simulating the Atmospheric Boundary Layer*, M.Sc. Thesis, San Jose State University.

Irwin, H.P.A.H., (1981) *The Design of Spires for Wind Simulation,* Journal of Wind Engineering and Industrial Aerodynamics 7:361-366.

Li, L.L., Zhang, L., Li, Q., Peng, R., Zhao, L., Zhang Y.F., Meng, Q.L., (2018) *CFD Study on the Characteristics of Temperature and Velocity in an Environmental Wind Tunnel*, 4th International Conference on Building Energy Environment, Melbourne, Australia.

Menter, F.R., (1994) *Two-Equation Eddy-Viscosity Turbulence Models for Engineering Applications*, AIAA Journal (32) 8:1598–1605.

Numeca CFD FINE/OPEN Theory Guide, (2010).

Pang J., Lin Z., (2008) *Development and Experimental Study on the Active Simulation Device in Boundary Layer Wind Tunnel*, Journal of Experiments in Fluid Mechanics (22) No. 3.

Shojaee, M.N., Kayışoğlu, B., Uzol, O., Kurç, Ö., (2009) *Design and Analysis of Passive Devices for Atmospheric Boundary Layer Simulation in a Short Aeronautical Wind Tunnel*, 5thAnkara International Aerospace Conference (AIAC), METU, Turkey.

Shojaee, M.N., Uzol, O., Kurç, Ö., (2014) *Atmospheric Boundary Layer Simulation in a Short Wind Tunnel*, Int. J. Environ. Sci. Technol. 11:59-68.

SONG, P., (2017) *Simulation of Atmospheric Boundary Layer in an Open-Loop Wind Tunnel Using Spire-Roughness-Element Technique*, M.Sc. Thesis, University of Windsor.

Yassen, Y.S., Abdelhamed A.S., (2015) *CFD Modeling of the Atmospheric Boundary Layer in Short Test Section Wind Tunnel,* American Journal of Aerospace Engineering. Vol.2, No.1-1, pp. 38-46.