

Turbulent Inlet boundary conditions for large eddy simulation

Vita, Giulio; Hemida, Hassan; Chaudhari, Ashvinkumar; Baniotopoulos, Charalampos

License:

None: All rights reserved

Document Version

Peer reviewed version

Citation for published version (Harvard):

Vita, G, Hemida, H, Chaudhari, A & Baniotopoulos, C 2018, Turbulent Inlet boundary conditions for large eddy simulation: an urban wind energy test-case. in C Baniotopoulos, C Borri, E Marino, B Blocken, H Hemida, M Veljkovic, T Morbiato, R Borg & N Hamza (eds), *Proceedings of WINERCOST'18 - Wind Energy Harvesting focusing on exploitation of the Mediterranean Sea.*, TS4.1.4, WINERCOST, Catanzaro, IT, pp. 253-260, WINERCOST'18 2nd International Conference on Wind Energy Harvesting, Catanzaro, Catanzaro, Italy, 21/03/18.

[Link to publication on Research at Birmingham portal](#)

Publisher Rights Statement:

Checked for eligibility: 28/03/2019

General rights

Unless a licence is specified above, all rights (including copyright and moral rights) in this document are retained by the authors and/or the copyright holders. The express permission of the copyright holder must be obtained for any use of this material other than for purposes permitted by law.

- Users may freely distribute the URL that is used to identify this publication.
- Users may download and/or print one copy of the publication from the University of Birmingham research portal for the purpose of private study or non-commercial research.
- User may use extracts from the document in line with the concept of 'fair dealing' under the Copyright, Designs and Patents Act 1988 (?)
- Users may not further distribute the material nor use it for the purposes of commercial gain.

Where a licence is displayed above, please note the terms and conditions of the licence govern your use of this document.

When citing, please reference the published version.

Take down policy

While the University of Birmingham exercises care and attention in making items available there are rare occasions when an item has been uploaded in error or has been deemed to be commercially or otherwise sensitive.

If you believe that this is the case for this document, please contact UBIRA@lists.bham.ac.uk providing details and we will remove access to the work immediately and investigate.

TURBULENT INLET BOUNDARY CONDITIONS FOR LARGE EDDY SIMULATION: A URBAN WIND ENERGY TEST-CASE

Giulio Vita^{1*}, Hassan Hemida¹, Ashvinkumar Chaudhari² and Charalampos Baniotopoulos¹

¹Dept. of Civil Engineering, School of Engineering, University of Birmingham, United Kingdom

²Centre of Computational Engineering and Integrated Design (CEID), Lappeenranta University of Technology, Finland

*Corresponding author: Giulio Vita, g.vita@bham.ac.uk

ABSTRACT

This paper reports on an investigation on numerical techniques to include turbulence at the inlet of Large Eddy Simulations (LES). Computational Fluid Dynamics (CFD) is promising as a technique to support wind tunnel results and provide a thorough understanding of the flow field. LES in particular may represent a breakthrough in the correct modelling of turbulence as found in complex geometrical configuration, such as in the urban environment. In fact, the poor understanding of local features of urban and buildings flows is the main reason of the lack of a convincing positioning strategy for Wind Energy Converters (WEC) in such conditions. Nevertheless, Wind Energy is considered a promising way of harvesting wind energy in built areas, as it slashes infrastructural wind energy costs. Developing a successful numerical technique would therefore improve our understanding of the inflow to be considered for urban wind turbines, help us model the effect of turbulence on the aerodynamic performance and eventually assess a positioning strategy which could in turn improve the current poor performance of this technology.

NOMENCLATURE

CFD	=	Computational Fluid Dynamics
FST	=	Free Stream Turbulence
LES	=	Large Eddy Simulation
SGS	=	Sub-Grid Scale model
WEC	=	Wind Energy Converter
WT	=	Wind Turbine

1. INTRODUCTION

Urban Wind Energy (UWE) is currently taking an ever-increasing role in the broad wind energy panorama [1]. Besides the debatable contribution to the energy demand or the matching between the energy consumption/production site, there are other reasons to further investigate UWE as a worthy niche of wind energy [2]. In fact, Small Wind Market is showing an exponential growth, involving numerous companies, investors and workplaces (Figure 1). Moreover, the unsuccessful UWE applications, as they are visible to the inhabitants of the urban environment affect the whole public perception and social acceptance of wind energy [3].

Unfortunately, many issues are responsible for the small or big failure of applications [4]. In particular, the aerodynamics of wind turbines (WTs) located in built areas is challenging and only few studies have shed light on how a WT responds to turbulent inflow structures [5]. For this reason, the positioning strategies for urban WTs are largely based on non-technical considerations: the classification of UWE applications is itself questionable [6].

In this work, a computational methodology based on Large Eddy Simulation is developed in order to include turbulent inlet boundary conditions to WTs and/or aerofoils simulations. A brief overview of existing techniques will be given, with reference to the two families of techniques: precursor simulation and synthetic turbulence methods.

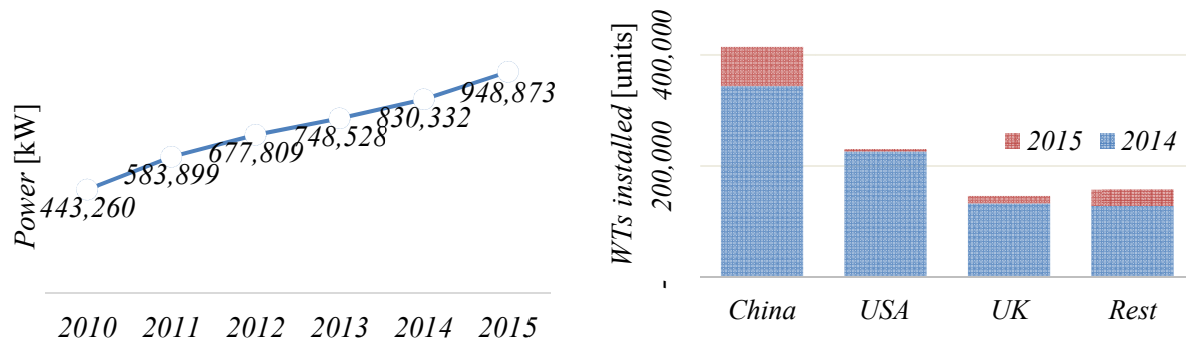


Figure 1. Cumulative Installed Power Capacity for Small WECs: 2016 Statistics ©WWEA [1].

2. METHODOLOGY

2.1. Experimental validation dataset

The experiment by Hemida et al. [7] provides the validation dataset for the numerical model. A series of wind tunnel experiments has been carried out at the Atmospheric Boundary Layer (ABL) Wind Tunnel Lab of the Ruhr-University of Bochum (RUB), within the scope of the COST-Action TU1304 WINERCOST. The RUB wind tunnel has a cross section of $1.6m \times 1.8m$ and a length of $9.4m$, in an open tunnel configuration with fan behind the test section (Fig.2a). The ABL is simulated equipping the wind tunnel inlet with a castellated barrier, turbulence generator fins, and roughness cubes (from $3.6cm$ to $1.6cm$) working as roughness elements (Fig.2c). The high-rise building model has a 1:300 scale, with a height-to-width ratio of $H/D=3$, where $H=400mm$ and $D=133.3mm$. Fig.2b shows the model mounted on the rotating test table of the wind tunnel.

Results include the velocity pattern above the rooftop, measured using a hot wire anemometer at different locations of the roof, schematised in Fig.2d) and surface pressure on the rooftop. However, this technique does not account for reversed flow, so results have been interpreted accordingly to detect separation. A specific focus has been given to turbulence intensity; it has been measured re. the velocity components in the y and z directions, u and w , for $z/D > 0.1$. Time histories of each signal have been obtained using a length window of $131s$, and all results are referred to the width of the model $D=133.3mm$ and the reference velocity $u_{ref}=u(z=H)=15.85m/s$ (Fig.2c).

The measurements of the velocity around the models have showed (as expected) that near the surface the flow is reversed, because of a separation bubble. Above $z/D \sim 0.3$, turbulence intensity declines to about 0.1 for both u and w , matching the one present in the upstream flow. This suggests that the building affects turbulence intensity up to a height of about one third of its width. Fig. 4c shows that wind velocity has the maximum increase of about 25% at about $z/D \sim 0.3$, at the centre of the roof (Pos.2). This is just above the shear layer between the separation region and the upstream flow, which is in agreement with previous experiments [8]. At the upstream edge (Pos. 4'), the maximum velocity occurs at a lower height, meaning that the it is maximised downstream to Pos.1. Beneath the maximum velocity area, the flow is highly turbulent. The turbulence intensity has its minimum value at the middle of the building, also for different directions of the wind. It has also been found that different direction yield separation cone vortices, which have a smaller size and height than the normal direction.

The inlet profile of mean velocity and turbulence intensity are shown in Fig.2c. Fig.2d shows the measurement for Pos. 1 and 2. These values provide a very useful validation dataset, as the flow field

is normally disregarded in studies about building aerodynamics in favour of the surface pressure field, which is more commonly considered.

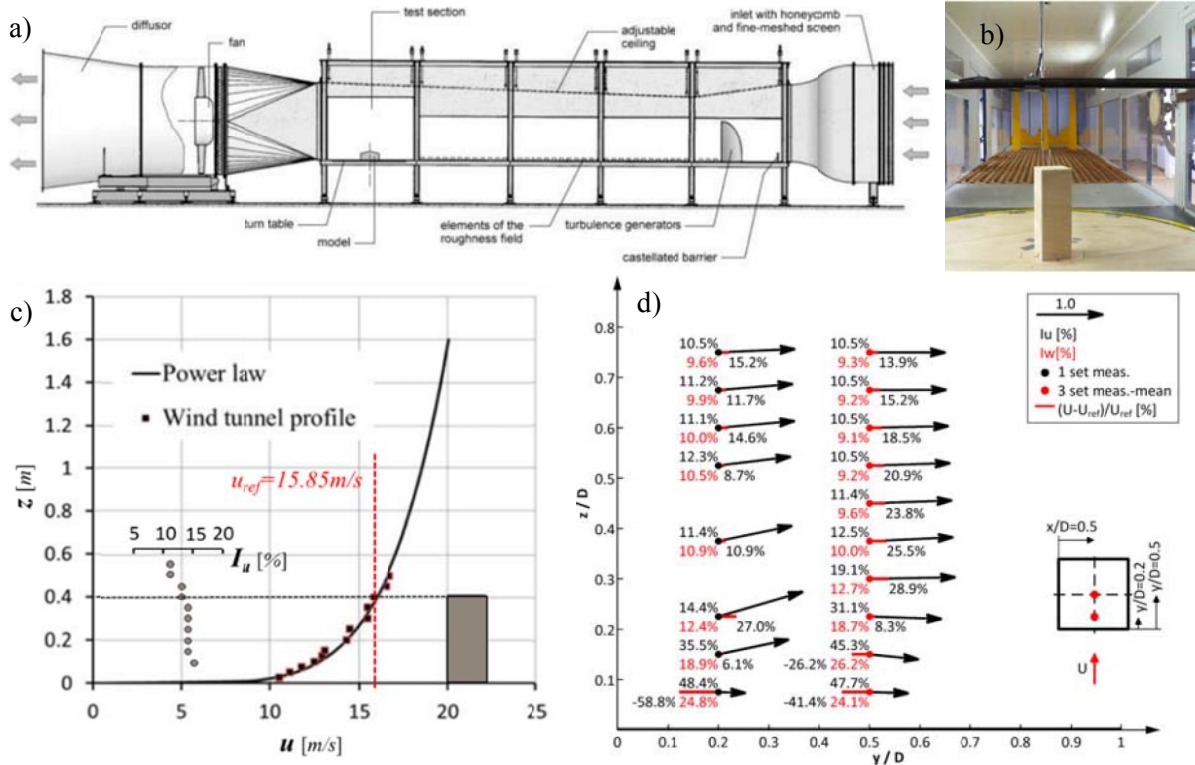


Figure 2. a) Wind Tunnel of the Ruhr-University Bochum; b) photo of the wind tunnel with the model mounted on a turning table; c) Wind-tunnel velocity profile ahead of the model; d) Velocity vectors, I_u and I_w as measured at pos. 1 and 2.

2.2. Numerical simulations

Large Eddy Simulation (LES) is a compromise between Direct Numerical Simulation (DNS) and Reynolds-Averaging of Navier Stokes equations (RANS). The first method computes all turbulent scales up to dissipation, while the latter models most of the turbulent scales using a suitable turbulence model. LES solves most of the turbulent scales, modelling a portion of them based on the definition of a filter, which in the large majority of models is defined on the size of the mesh cells. The sub-grid scale SGS model is defined on the concept that at small scales turbulence has roughly universal characteristics rather independent on the geometry considered [9].

A turbulent inlet can be generated in two ways. A way is to model the upstream part of the wind tunnel, which is responsible of the development of the ABL in the experiment (method A). This approach accounts as a sort of virtual wind tunnel to be modelled. However, a great amount of caution is needed for computational accuracy, as with such large domains and strict mesh requirements, the convergence of the simulation as well as their cost is at stake. A very promising alternative is to introduce noise in the equations by using appropriate time-varying boundary conditions (method B). Two general methodologies exist to generate fluctuating inlet conditions for LES: 1) precursor simulation and 2) synthetic turbulence methods [10]. These two approaches are complementary rather than alternative [11]. The precursor simulation methods rely on a previous simulation that gives a database of fluctuating conditions to be used as inlet for the LES model of interest, thus reproducing

inlet with the expected statistics of turbulence. Synthetic methods on the other hand improve convergence and accuracy without having a physical meaning for the inlet itself [12].

A steady state RANS (Reynolds Averaged Navier-Stokes) simulation has also been developed using the $k-\omega$ SST turbulence model, to get a brief idea of the flow pattern and check if the mesh requirements are met. The objective is to have a rough estimation of the turbulent quantities at stake. A preliminary comparison with the LES (Large Eddy Simulation) approach is then proposed, in order to discuss the importance of modelling the fluctuating statistics. In fact, even though the RANS approach has shown good viability for the estimation of the averaged fields, it is hindered by the modelling of fluctuating statistics [13, 14]

A block structured mesh has been constructed, having $y^+ \sim 1$ and choosing the blocking strategy in order to limit the along-flow size of the elements to respect the CFL condition which imposes $Co < 1$. This has brought to the choice of a time step of $\Delta t = 5e^{-5}$ for the LES model presented. The Smagorinsky-Lilly model has been chosen as Sub-Grid Scale model (SGS), with the use of the van Driest damping function¹, as implemented in the OpenFOAM[®] software v.2.3.1.

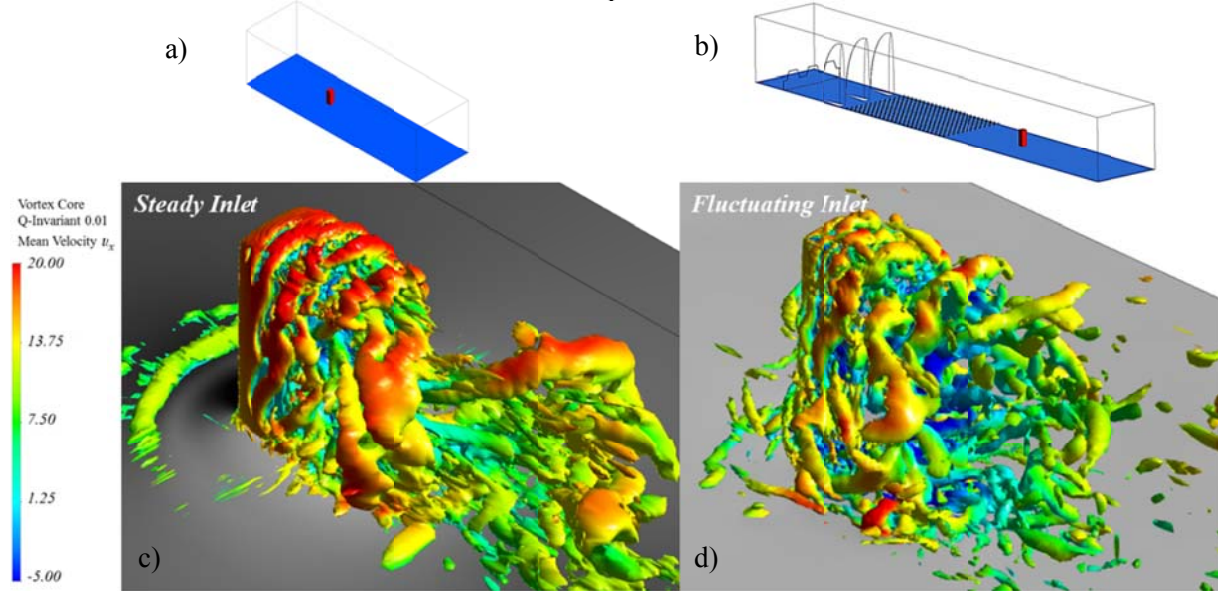


Figure 3. Fluctuating and steady inlet for LES simulation. a) and b) computational domain, c) and d) flow pattern in terms of velocity-invariant $Q=0.1$.

3. RESULTS AND DISCUSSION

These two methodologies have been used to model the effect of turbulence on WECs that is found at a suitable location for the positioning of WTs within the built environment, such as the high-rise building as described in Section 2.1. The aim is to understand the effect of the turbulent inlet conditions on the development of statistics at the location of interest which is the rood top, where eventually a WEC could be placed. For this preliminary study, two models have been developed to compare the performance of turbulent inlet generation against an LES model which considers the development of the flow and its characteristics. In this paper, the methodology (A) is compared with a

¹ A damping function, such as the one formulated by Van Driest $l_{sgs} = C_{sgs} \Delta \left(1 - e^{-y^+/A^+}\right)$, where $A^+ = 26$, lowers the value of the model constant C_{sgs} or C_s in those regions of the flow where viscosity effects become preponderant. In fact, the Smagorinsky model uses a fixed constant to take into account the SGS effects, potentially yielding an unphysical behaviour.

The eddy viscosity reads $\nu_{sgs} = (C_{sgs} \Delta)^2 |\tilde{S}| = (C_{sgs} \Delta)^2 \sqrt{2 \tilde{S}_{ij} \tilde{S}_{ij}}$ where the Smagorinsky model constant has been set to $C_s = 0.17$.

steady state case, where a laminar inlet boundary layer profile is imposed at the inlet. The two computational domains are shown in Fig. 3a and b respectively. The flow field is indicated as steady and fluctuating. From a qualitative assessment of results presented in Fig. 3, it can be argued that the flow field looks less organised in the fluctuating case, as many flow structures from upstream roughness elements contribute to the break of vortices, rather than the separated flow due to the bluff body. However, the shedding of the vortices, i.e. the frequency at which vortices are shed from the separation corners of the building, can be considered basically unchanged. This can be quickly assessed by considering the iso-vorticity map, where the vortex cores are roughly placed at the same distance in both cases.

Arguably, for wind energy purposes, the two cases provide a similar flow pattern, which in turn can represent a major simplification of a possible methodology of introducing turbulence effects on WEC performance, when placed close to a bluff body.

A better estimation of the accuracy of the two simulation is given in Fig. 4, where the steady and fluctuating case are compared with the RANS simulation for the most common turbulence statistics.

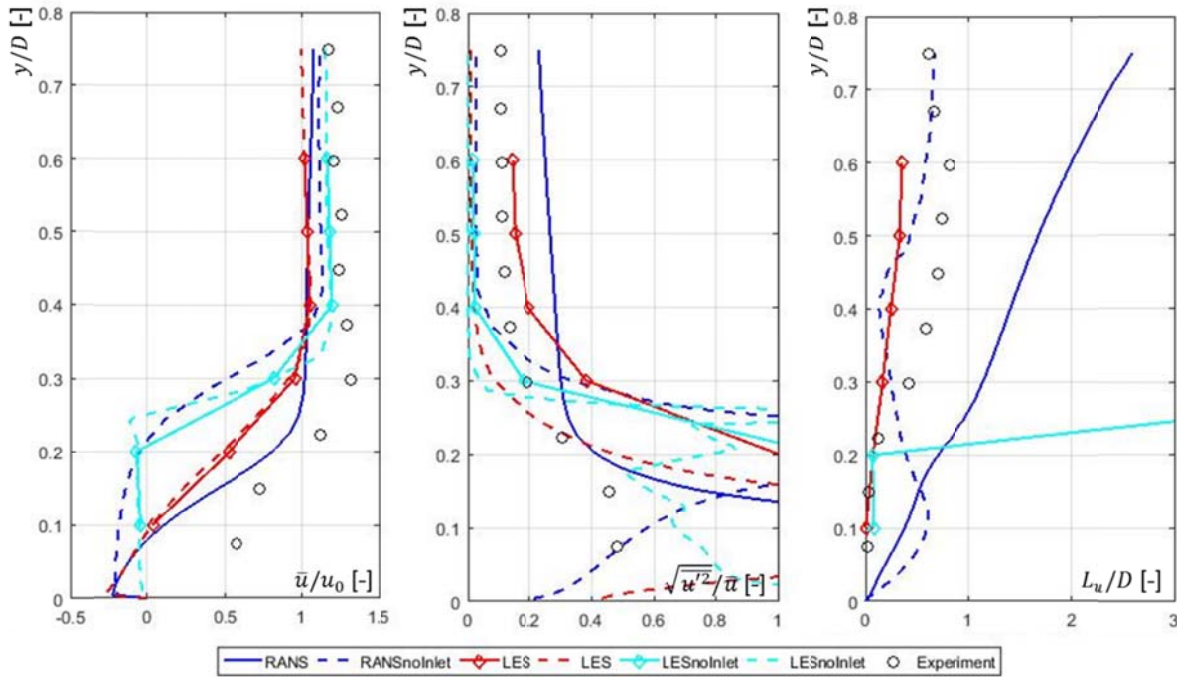


Figure 4. Respectively Average Velocity, Turbulence intensity and integral length scale of the high-rise building testcase with comparison of inlet geometry and no inlet for both RANS and LES.

The LES simulation with fluctuating inlet is the closest to the experimental data in modelling turbulence intensity and integral length scale. Not surprisingly, the RANS model performs better in modelling the mean flow. It is rather evident how the steady inlet case does not capture the physics of the flow. However, a closer look to the measured signals clarifies the mismatch between the different techniques. In Fig. 5 the experimental data is plotted together with the steady and fluctuating inlet simulations. Due to the preliminary nature of the simulations it is evident how the fluctuating signal matches peaks of the fluctuating one, while the steady one is rather damped and much shorter. This is also confirmed by the spectra, which are plotted in Fig. 6 for positions 2 and 3 at two different heights. The spectra overlap quite well at lower frequencies, while they diverge at the highest. This is also due to the difference in the length of the signal and in the nature of the chosen sub-grid scale model, which interacts with the solved vortex structures.

LES simulation, although potentially powerful in producing highly accurate data, has still to become viable due to the practicality of the LES approach. That is why the usage of boundary condition fluctuating techniques might be the forthcoming solution to the practical LES usage.

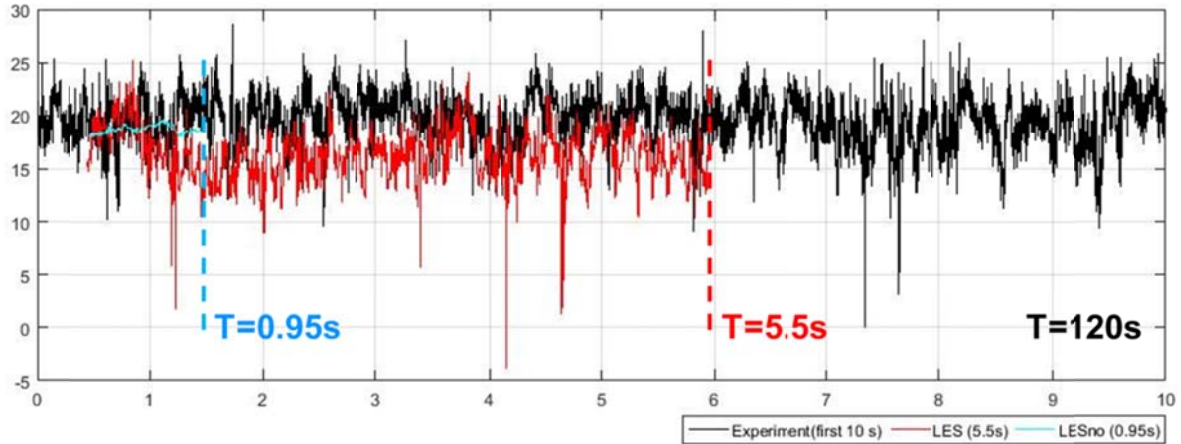


Figure 5 – Time histories of velocity signal for experiment, steady and fluctuating inlet.

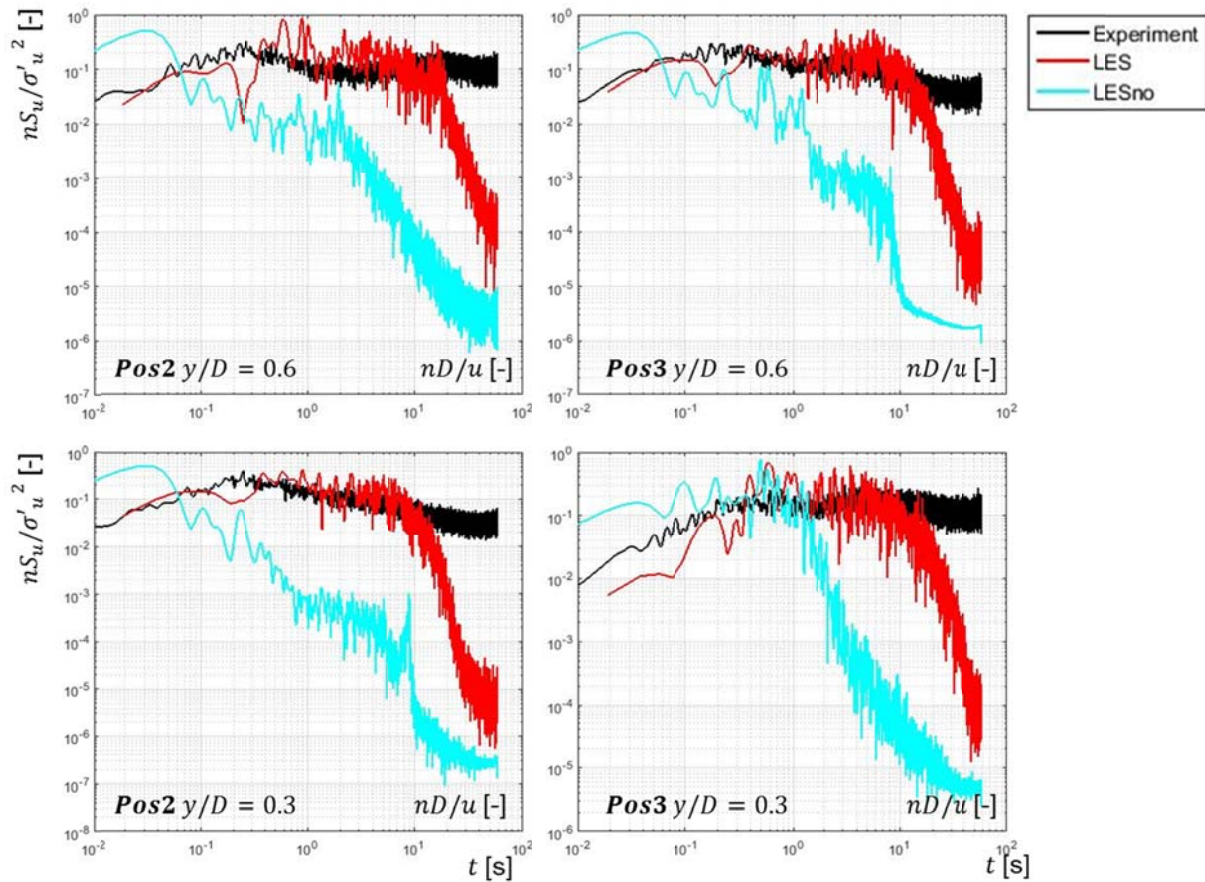


Figure 6 – Energy spectra at different positions and heights.

A methodology based on turbulent inlet generation is expected to greatly improve accuracy and reliability of results and provide a powerful tool for the optimization of wind energy harvesting in the urban environment and further engineering applications which require a thorough understanding of the

inflow. However, the physical significance of results has to be ascertained. That is why an STSM has been organised in collaboration with the Centre of Computational Engineering and Integrated Design (CEID) of the Lappeenranta University of Technology (LUT) with the support of Dr. Ashvinkumar Chaudhari. The STSM has the purpose of investigating the possibilities implemented in OpenFOAM to introduce numerical noise with statistics having physical meaning at the inlet of LES. Over the duration of the STSM several OpenFOAM tutorials have been developed. Results will be compared to the modelled LES high-rise building to understand the limitation of the simplified approach offered by a variety of turbulent inlet conditions. A reference box case, comprising of a grid inlet, is considered. The generation of virtual grid turbulence has been implemented splitting the inlet boundary into a wall and an inlet region respectively. In this way turbulence is produced at the interface between the wall boundary and the inlet. It is found in literature that this technique reproduces grid generated turbulence virtually [15] with a good degree of approximation. However, it is extremely important to choose periodic boundary conditions at the boundaries of the box, which in turn needs to have a specific size and mesh resolution. The same computational domain and mesh is then used to include turbulent inlet synthetic turbulence. The turbulentInlet random fluctuations generator is used in comparison with two techniques developed respectively at the ETH [16] and found in the github repository [17].

In Figure 1, the flow field as computed using the inlet virtual grid and the random turbulentInlet is shown. The numerical noise introduced using random fluctuation with a given amplitude is responsible for the higher maximum velocity of Fig. 7a, while in Fig 7b the maximum velocity corresponds to the inlet velocity, calculated considering the porosity of the virtual grid. In Fig. 8, the Energy spectrum of the longitudinal component of the velocity field is plotted for the two cases. It is evident how the inertial range region of the spectrum does not develop, hence raising doubts on the validity of the boundary condition. At the high end of the spectrum, the behaviour follows the $-5/3$ law, as for the smallest scales the sub grid scale model is responsible for the resulting flow field. In this preliminary account of results, one can conclude that the generation of virtual turbulence with given statistics requires a great deal of attention, as results strongly vary with the size of the domain, the resolution of the mesh and the duration of the simulation.

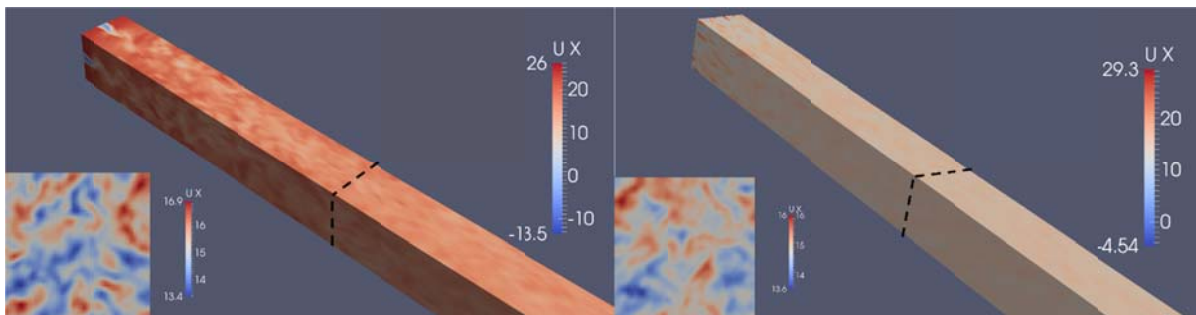


Figure 7 – a) Turbulence box generated with virtual grid and b) random fluctuation generator, with detail of instantaneous velocity field.

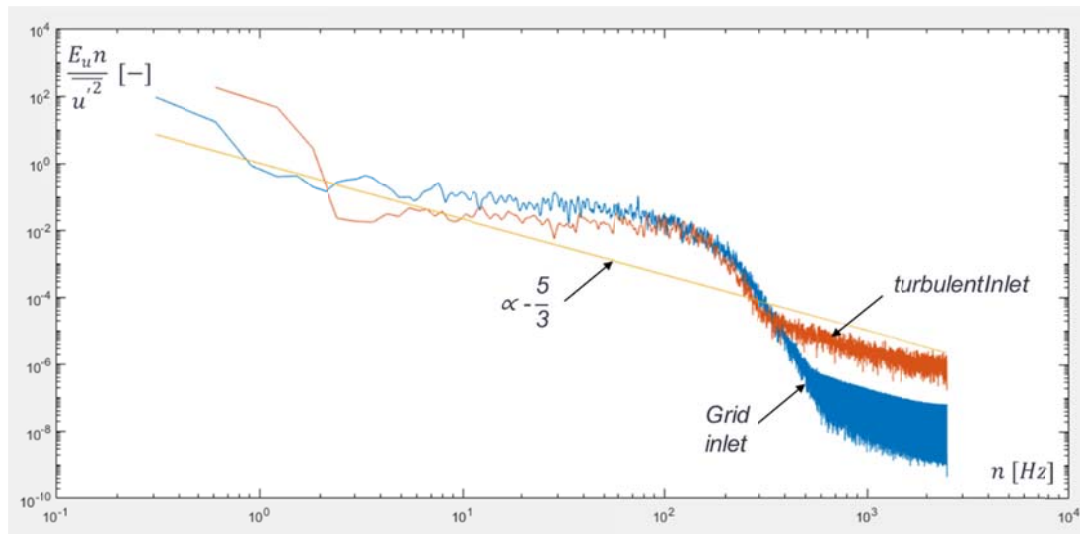


Figure 8 – Energy Spectrum for different turbulent inlet generation techniques with -5/3 law comparison.

Results gained using the turbulent inlet techniques will be compared with actual physical results from the LES simulation of the wind tunnel setup. The aim is to provide a suitable simple and fast technique to generate a fluctuating turbulence field having certain turbulence characteristics as in the experiment. It is questionable whether the inclusion of random fluctuations at the inlet of the steady simulation would help the convergence and robustness of the LES simulation towards the experimental results. It will be the topic of further research from the authors to find out.

ACKNOWLEDGEMENTS

The authors acknowledge with thanks the support of the European Commission Framework Program “Horizon 2020”, through the Marie Skłodowska-Curie Innovative Training Networks (ITN) “AEOLUS4FUTURE - Efficient harvesting of the wind energy” (H2020-MSCA-ITN-2014: Grant agreement no. 643167), to the present research project. The authors also acknowledge with thanks the WINERCOST TU1304 COST Action for having provided them with a plethora of collaboration activities related to small wind technology.

REFERENCES

- [1] Gsänger, S., & Pitteloud, J.-D. (2015). Small Wind World Report 2015. Retrieved from www.wwindea.org
- [2] Enevoldsen, P., & Sovacool, B. K. (2016). Examining the social acceptance of wind energy: Practical guidelines for onshore wind project development in France. *Renewable and Sustainable Energy Reviews*, 53, 178–184. <https://doi.org/10.1016/j.rser.2015.08.041>
- [3] Hamza, Neveen. (2015). URBAN WIND ENERGY EXPOSING SUSTAINABILITY SYMBOLISM WINERCOST Workshop ‘Trends and Challenges for Wind Energy Harvesting’, Coimbra, 30-31 March 2015.
- [4] Balduzzi, F., Bianchini, A., & Ferrari, L. (2012). Microeolic turbines in the built environment: Influence of the installation site on the potential energy yield. *Renewable Energy*, 45, 163–174. <https://doi.org/10.1016/j.renene.2012.02.022>

- [5] Devinant, P., Laverne, T., & Hureau, J. (2002). Experimental study of wind-turbine airfoil aerodynamics in high turbulence. *Journal of Wind Engineering and Industrial Aerodynamics*, 90(6), 689–707. [https://doi.org/10.1016/S0167-6105\(02\)00162-9](https://doi.org/10.1016/S0167-6105(02)00162-9)
- [6] Walker, S. L. (2011). Building mounted wind turbines and their suitability for the urban scale—A review of methods of estimating urban wind resource. *Energy and Buildings*, 43(8), 1852–1862. <https://doi.org/10.1016/j.enbuild.2011.03.032>
- [7] Hemida, H., Šarkic, A., Gillmeier, S., Höffer, R., Experimental investigation of wind flow above the roof of high-rise building, WINERCOST Workshop "Trends and Challenges for Wind Energy Harvesting", Coimbra, Portugal, 30-31 March 2015, p.25-34.
- [8] H. Hemida, "Large-eddy simulation of the above roof flow of a high-rise building for micro-wind turbines," in 11th UK Conference on Wind Engineering, 2014.
- [9] Pope, S. B. (2000). *Turbulent Flows*. Cambridge University Press (Vol. 1). <https://doi.org/10.1088/1468-5248/1/1/702>
- [10] Tabor, G. R., & Baba-Ahmadi, M. H. (2010). Inlet conditions for large eddy simulation: A review. *Computers & Fluids*, 39(4), 553–567. <https://doi.org/10.1016/j.compfluid.2009.10.007>
- [11] Lund, T. S., Wu, X., & Squires, K. D. (1998). Generation of Turbulent Inflow Data for Spatially-Developing Boundary Layer Simulations. *JOURNAL OF COMPUTATIONAL PHYSICS*, 140, 233–258.
- [12] Chaudhari, A. (2014). Large-eddy simulation of wind flows over complex terrains for wind energy applications. Lappeenranta University of Technology.
- [13] B. Blocken, T. Stathopoulos, and J. P. A. J. van Beeck, "Pedestrian-level wind conditions around buildings: Review of wind-tunnel and CFD techniques and their accuracy for wind comfort assessment," *Build. Environ.*, vol. 100, pp. 50–81, 2016.
- [14] Ozmen, Y., Baydar, E., & van Beeck, J. P. A. J. (2016). Wind flow over the low-rise building models with gabled roofs having different pitch angles. *Building and Environment*, 95, 63–74. <https://doi.org/10.1016/j.buildenv.2015.09.014>
- [15] T. Blackmore, W. M. J. Batten, and A. S. Bahaj, "Inlet grid-generated turbulence for large-eddy simulations," *Int. J. Comput. Fluid Dyn.*, vol. 27, no. 6–7, pp. 307–315, Jul. 2013.
- [16] M. C. Immer, "Time-resolved measurement and simulation of local scale turbulent urban flow," 2016.
- [17] E. Asgari and M. Tadjfar, "Assessment of four inflow conditions on large-eddy simulation of a gently curved backward-facing step," *J. Turbul.*, vol. 18, no. 1, pp. 61–86, Jan. 2017.