WIND FLOW RESOURCE ANALYSIS OF URBAN STRUCTURES, A VALIDATION STUDY

Aya Aihara[1], Bahri Uzunoğlu[2], Anders Goude[3]

Department of Engineering Sciences, Division of Electricity,
Centre for Renewable Electric Energy Conversion,
Uppsala University, The Ångström Laboratory

ABSTRACT

In order to have better insight into the physics of the urban wind turbines, a Computational Fluid Dynamics (CFD) flow solver has been developed for industrial applications by Uppsala University and SOLUTE Ingenieros. Urban wind resource assessment for small scale wind applications present several challenges and complexities for that are different from large-scale wind power generation. Urban boundary layer relevant in this regime of flows have different horizontal profiles impacted by the buildings, low speed wind regimes, separation and different turbulence characteristics. Preliminary measurement results will be presented for a particular site in Huesca Spain where a measurement campaign is undertaken to validate the CFD results.

INTRODUCTION

The small scale wind mainly corresponds to turbines installed in rural and isolated areas. As 80 % of European population lives in cities and the EU Directive 2010/31/EU on Energy Performance of Buildings requires that “Member States shall ensure that by 31 December 2020 all new buildings are nearly zero-energy buildings”. This is a commercial opportunity while also provides a motivation to investigate technical challenges related to the peculiarities of Urban wind regime. Urban wind resource assessment for small scale wind applications present several challenges and complexities for that are different from large-scale wind power generation [1].

Urban boundary layer relevant to this kind of flows have different horizontal profiles impacted by the buildings, low speed wind regimes, separation and different turbulence characteristics. Urban wind regimes that are relevant to urban boundary layer research investigates wind profiles and thermally driven secondary circulations over cities. The urban-roughness layer profile, which is the most relevant profile for wind installations, can be approximated by profile laws using logarithmic wind profile for neutral, stable and unstable stratification. General formulations for flat terrain have been noted to be not accurate for complex flows like urban flows. Therefore, we have developed a graphical user interface (GUI) for industrial applications that employs an industry based CFD solver namely OpenFOAM [1].

In order to have better insight into the physics of the urban wind turbines, an European Framework project with acronym WINDUR has been undertaken. As part of this work, the complexity of the problem motivated us to look at the physics of Urban flow problem first by measurements on several sites in Spain based on different climate classification regions defined. The preliminary results of the measurement campaign will be presented for a site in Huesca Spain to validate the CFD results with also preliminary results [1].

THEORY

Since the general formulations for flat terrain is not suitable to accurately express the urban flows, the following approximation can be used for the Prandtl layer.

$$\boldsymbol{U} = \frac{\boldsymbol{U}^*}{\kappa} \boldsymbol{ln} \frac{\boldsymbol{z} - \boldsymbol{d}}{\boldsymbol{z_0}}$$

[1] aya.aihara@angstrom.uu.se
[2] bahri.uzunoglu@angstrom.uu.se
[3] anders.goude@angstrom.uu.se

where $\boldsymbol{U^*}$ is characteristic velocity and κ is von karman constant, which is 0.41, $\boldsymbol{z_0}$ is roughness length, $\boldsymbol{d}$ is displacement height, which gives the vertical displacement of the flow for buildings [2][3][4][7][8]. The vertical height is denoted by $\boldsymbol{z}$. This relation has been modified for the second layer which is wake layer with new parameter $\boldsymbol{\alpha}$ to reflect the impact of buildings

$$\boldsymbol{U = \alpha \frac{U^*}{\kappa} ln \frac{z-d}{z_0}}$$

This was further modified for the first layer as an exponential rule for the bottom urban canopy layer (UCL)

$$\boldsymbol{U = U_h \exp\left(a\left(\frac{z}{h}-1\right)\right)}$$

where $\boldsymbol{a}$ is a constant which is dependent on building morphology, $\boldsymbol{h}$ is the height of the building and $\boldsymbol{U_h}$ is the velocity at the building height. The detail explanation can be found in [2] [3] [4].

For CFD solutions, steady state equations of fluid mechanics will be used so all the solutions are time averaged as discussed in reference [1]. Thus, Reynolds Averaged Navier Stokes (RANS) equations are implemented. RANS comparison to Large Eddy Simulation (LES) is computationally less demanding [9][10][11]. For incompressible flows, the general form of the Navier-Stokes is given by

$$\boldsymbol{\frac{\partial}{\partial x_j}[\rho u_j] = 0, \qquad \frac{\partial}{\partial t}(\rho u_i) + \frac{\partial}{\partial x_j}[\rho u_i u_j + p\delta_{ij} - \tau_{ij}] = 0 \qquad (i = 1, 2, 3)}$$

where $\boldsymbol{\rho}$ defines density, $\boldsymbol{p}$ defines hydrostatic pressure, $\boldsymbol{u_j}$ defines velocity $[\boldsymbol{u, v, w}]$ and $\boldsymbol{x_j}$ spatial coordinates and the stress tensor $\boldsymbol{\tau_{ji}}$ depends linearly on the rate-of-strain tensor $\boldsymbol{S_{ij}}$ and dynamic viscosity $\boldsymbol{\mu}$. In an appropriate time $\boldsymbol{T}$ interval if the velocity $\mathbf{u}$ is time averaged:

$$\bar{\mathbf{u}}(\mathbf{x}) = \frac{1}{T}\int_{t_0}^{t_0+T} \mathbf{u}(\mathbf{x}, \mathbf{t})\mathbf{dt}$$

The averaged term and the fluctuation term of the Reynolds decomposition of velocity can simplify convective term as follows

$$\boldsymbol{\frac{\partial}{\partial x_j}[\rho \overline{u_j}] = 0, \qquad \frac{\partial}{\partial t}(\rho \overline{u_i}) + \frac{\partial}{\partial x_j}[\rho \overline{u_i}\,\overline{u_j} + \bar{p}\delta_{ij} - \overline{\tau_{ij}} + \tau_{ij}^R] = 0 \qquad (i = 1, 2, 3)}$$

which is Reynolds Averaged Navier-Stokes equation (RANS) [5].

RESULTS

There are two measurement stations in Huesca site for validation purposes, and the orography and the buildings involved for numerical simulation are displayed in Figure 1. Thus two masts were placed at 3.5 m height from the roof of the buildings, namely EDIF 3 and EDIF 4. The measurements were recorded for every 10 minute interval.

Figure 1. Huesca measurement site in Spain (left) and wind mast (right)

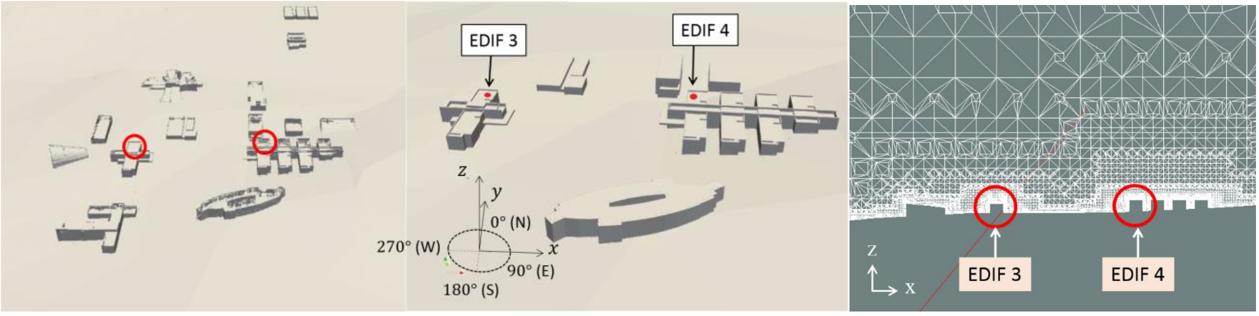

Figure 2. Computational mesh of building in Huesca, EDIF 3 and EDIF 4 developed in graphical user interface software

Based on the OpenFOAM solver, a simulation tool has been developed for the purpose of simulating in urban environments. The user can input the geometry, such as buildings and surface roughness and also can change parameters used for the mesh generation as well as boundary conditions. The generated mesh of the buildings in Huesca site is shown in Figure 2.

The wind speed at the height of the mast placed on the roof was extracted from the simulation results. The logarithmic boundary layer is used for the inlet velocity as the boundary conditions, and the flow velocity at 1000 m height was set to 10m/s. The simulations are executed for 7.5 degree sectors. Figure 3 shows the wind rose for the mean wind speed. It indicates that the simulation result represents the influence of the surrounding buildings as can be seen in the measurement that wind speed is highly dependent on the wind direction.

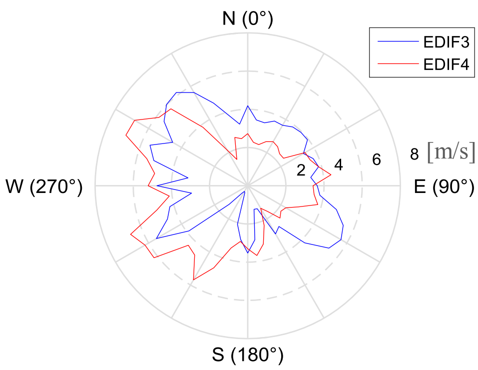

Figure 3. Mean wind speed of EDIF 3 and EDIF 4 building at each sector simulated

The result of the simulation is evaluated by comparing with the measurement data. They are validated at two characteristic sectors, 90 and 285 degree where the measurement result shows that the wind is especially highly distributed.

Here the linear relationship between the wind speed at EDIF 3 and EDIF 4 is compared as shown in Table 1. As a result, it can be said that the CFD simulation shows good results since it captures the characteristics of urban flows well as expected. When the wind speed at EDIF 3 and EDIF 4 are denoted as U_x and U_y, the linear relationship of each sector are determined as follows. For the simulation data, the ratio C_s is obtained by calculating U_x/U_y. On the other hand, the linear correlation is applied to the measurement data so that they are expressed by the equation $U_x = a \cdot U_y + b$. Here the slope a is defined as C_m, while C_{m0} is the slope of the equation fitted through the origin ($b = 0$). Even though the masts are put near the roof of the building and higher turbulence could be involved, the simulation results suggest the values close to the measurement results. It will be worth testing more simulations with different mesh and boundary conditions, and the validation should also be done for other sectors to confirm the result.

Table 1. Comparison between the simulation and measurement result with regards to linear relationship between wind speed at EDIF 3 and EDIF 4 at sector 90 and 285 degree.

		90 degree	285 degree
Simulation	C_s	1.064	0.9302
Measurement	C_m	1.129	0.6939
	C_{m0}	1.418	0.7660

CONCLUSIONS

In this study, a CFD flow solver has been implemented for small scale wind applications. A measurement campaign, as part of the WINDUR EU Framework 7 project, was undertaken in Huesca site of Spain in order to validate the CFD results. The results are validated employing two point observations at the same site. The simulation results from one measurements to other measurement mast are cross checked and validated. The preliminary results are presented.

REFERENCES

[1] A. Goude, B. Uzunoglu, G. Giovannini, J. Magdalena and A. Fernandez, “A GUI for Urban Wind Flow CFD Analysis of Small Scale Wind Applications”, 2015 International Conference on Cyberworlds (CW), pp. 193-199, 2015.

[2] Malcolm A. Heath, John D. Walshe, and Simon J. Watson, “Estimating the potential yield of small building mounted wind turbines”, Wind Energy, 10(3), pp. 271–287, 2007.

[3] Jian-Zhong, Li Hui-Jun, and Zhang Kai, “New expressions for the surface roughness length and displacement height in the atmospheric boundary layer”, Chinese Physics, 16(7), pp. 2033-2039, 2007.

[4] Walker, Sara Louise, “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, pp. 1852-1862, 2011.

[5] H. Versteeg and W. Malalasekera, “An Introduction to Computational Fluid Dynamics: The Finite Volume ethod”, Pearson Education Limited, US, 2007.

[6] Report from WINDUR project of the 7th EU Framwork Programme. “An urban boundary layer wind measurement campaign in Spain based on Köppen-Geiger climate classification”, http://www.project-windur.eu/

[7] Daniel R. Drew, Janet F. Barlow, and Sin E. Lane, “Observations of wind speed profiles over greater london, uk, using a doppler lidar”, Journal of Wind Engineering and Industrial Aerodynamics, 121(0):98 – 105, 2013.

[8] S. Emeis. “Wind Energy Meteorology: Atmospheric Physics for Wind Power Generation. Green Energy and Technology”, Springer Berlin Heidelberg, 2012.

[9] Jae-Jin Kim and Jong-Jin Baik, “A numerical study of the effects of ambient wind direction on flow and dispersion in urban street canyons using the RNG k-ε turbulence model”, Atmospheric Environment, 38(19):3039 – 3048, 2004.

[10] M. Tutar and G. Oguz, “Computational modeling of wind flow around a group of buildings”, International Journal of Computational Fluid Dynamics, 18(8):651–670, December 2004.

[11] Aishe Zhang, Cuilan Gao, and Ling Zhang, “Numerical simulation of the wind field around different building arrangements”, Journal of Wind Engineering and Industrial Aerodynamics, 93(12):891 – 904, 2005.