Available online at www.sciencedirect.com

ScienceDirect

Energy Procedia 134 (2017) 366–375

Procedia Engineering

www.elsevier.com/locate/procedia

9th International Conference on Sustainability in Energy and Buildings, SEB-17, 5-7 July 2017, Chania, Crete, Greece

Performance of Detached Eddy Simulation applied to Analysis of a University Campus Wind Environment

Rallou Dadioti[a], Simon Rees[b]

[a]*Institute of Energy and Sustainable Development, De Montfort University, Leicester, LE1 9BH, UK*
[b]*School of Civil Engineering, University of Leeds, LS2 9JT, UK*

Abstract

This paper investigates the performance of a Detached Eddy Simulation (DES) turbulence modelling approach implemented in the open-source CFD library OpenFOAM to analysis of external flows in urban environments in the context of micro wind turbine applications. For this purpose, we examine the predicted flows at the De Montfort university (DMU) campus in Leicester, UK and make comparisons with high frequency wind data recorded by three ultrasonic anemometers. We find that the CFD results using the DES model are in good agreement with the anemometer data in terms of both relative wind speeds and directions. We conclude that DES offers robustness and accuracy over a range of wind conditions and is well suited to analysis of wind energy potentials in complex urban environments.

Peer-review under responsibility of KES International.

Keywords: CFD;urban environment;wind engineering;ultrasonic anemometers

---

1. Introduction

In this study the analysis of wind flow at a university campus is based on numerical modelling using Computational Fluid Dynamics (CFD) approaches. The main aim is to evaluate the accuracy of CFD modelling applied to external flows in complex urban environments where turbulence is treated using a Detached Eddy Simulation (DES) model. This approach allows the large-scale features in the wake regions near buildings to be well resolved but avoids the high computational cost of fully resolving near-wall regions. It is these wake regions that are of most interest in the context of evaluating wind energy potentials in urban environments.

In the following sections, we comment on relevant literature in the field of CFD modeling of building external flows, the methodology applied in this study is described and results are presented. The discussion section offers validation evidence for DES approaches and highlights areas for further investigation.

---

[*] Corresponding author. Tel.: +30-697-633-7793.
E-mail address: rallou.dadioti@gmail.com

Peer-review under responsibility of KES International.
10.1016/j.egypro.2017.09.551

Nomenclature

CAD	Computer-aided design
CFD	Computational Fluid Dynamics
DES	Detached Eddy Simulation
DMU	De Montfort University
EM	Edith Murphy
H	Maximum height
ITCB	Integrated Technology Complex building
KL	Kimberlin Library
LES	Large Eddy Simulation
QB	Queens Building
RANS	Reynolds Averaged Navier Stokes
SA	Spalart Allmaras
SM	Short Measurements

2. CFD applied to building external flows

2.1. Wind flow analysis in urban areas

The motivation for the study and developing new analysis approaches has been to examine the potential for micro wind turbine applications in complex urban environments. Environments such as the city-centre campus studied, are challenging due to the interaction between the flows around neighbouring buildings and for the potential for one building (or building feature) to create a ’wind shadow’ over an adjacent space or building. A number of approaches to this type of study are possible. Analytical methods do not account for the impact of individual obstacles on the flow and hence cannot predict the variability of the wind speed in the vicinity to the buildings [1]. ‘Wind speeds at sheltered sites are difficult to predict accurately without site specific fluid dynamical modelling’, as reported by Millward-Hopkins et al.[2]. Analysis of on-site measurements can be considered the most accurate approach. However, such measurements are costly and time-consuming and they can only capture the wind characteristics at discrete points [3]. Statistical analysis (e.g. Weibull analysis) can contribute to macro-siting studies, however, it cannot provide the precision required for micro-siting [4].

Computational Fluid Dynamics (CFD) is a numerical analysis approach which has been shown to be a promising technique to analyse the wind flow in complex urban environments for evaluating the potential for micro wind turbines installation [4]. According to Karthikeya et al.[5] ‘A CFD simulation approach would be ideal to understand the actual flow patterns in these localities to decide the best locations for wind turbine installations’. It has the advantages of providing information over the whole of the domain of interest and can be more economical than measurement campaigns. The ability to finely discriminate between locations (i.e. potential turbine sites) is particularly important in geometrically complex urban environments. Key questions in applying CFD methods are the approach taken to modelling turbulence and the computational costs and we seek to address these issues here.

2.2. CFD studies in real urban environments

CFD studies of complex urban areas validated using field measurements are extremely limited in number. Tabrizi et al.[6] used the ANSYS-CFX software to model an area in Port-Kennedy, Western Australia. He used the Reynolds averaged Navier-Stokes (RANS) approach to find the time-averaged flow field using a k-ϵ turbulence model and the results were compared with experimental data. Yang et al.[4] carried out simulations of the wind flow over the ITCB building on the National Taipei University of Technology campus, using the ANSYS-Fluent software. He used RANS equations with the Realizable k-ϵ turbulence model and verified the results against on-site data. Kalmikov et al.[7] considered the complex geometry of the Massachusetts Institute of Technology campus in the USA in their CFD study. In their work the 'UrbaWind' software solved the RANS equations with the k-L model for the turbulent fluxes and local wind measurements were integrated for validating the model.

2.3. CFD approaches

Even the most sophisticated RANS turbulence models are not able to fully represent the flows dominated by large-scale features such as flows around buildings in urban environments [8,9]. Cheng et al.[10] found that the RANS results were considerably different from the Large Eddy Simulation (LES) when compared with the experimental data [11]. LES calculations are fully transient such that the flow field is calculated at a frequency that allows large scale turbulent features to be fully resolved (time averaged values are calculated as a post processing exercise) but requires very fine meshes and very short time-steps. Cheng et al.[10] also compared the RANS approach with LES and concluded that RANS modelling gives significant uncertainties in description of unsteady phenomena; complex features such as separation zones, vortex shedding and recirculation zones (Fig. 1) were better reproduced with LES than with RANS calculations [12].

However, the simulation of atmospheric high Reynolds number flows using full LES approaches is very expensive in terms of computational resources [8] and so another numerical model is required to deal with these limitations. Villiers[13] has suggested 'In aerodynamics and bluff body flows, where the boundary layer is small and the flows are dominated by large length scales and separation induced vortices, the DES model is anticipated to produce LES quality solutions at reduced costs in terms of computational resources'. Consequently this study has used a hybrid DES method [14] which uses RANS turbulence modelling in the boundary layer and employs LES in separated regions. This offers improved prediction of flows compared to RANS methods but is less computationally demanding than full transient LES approaches [8,15].

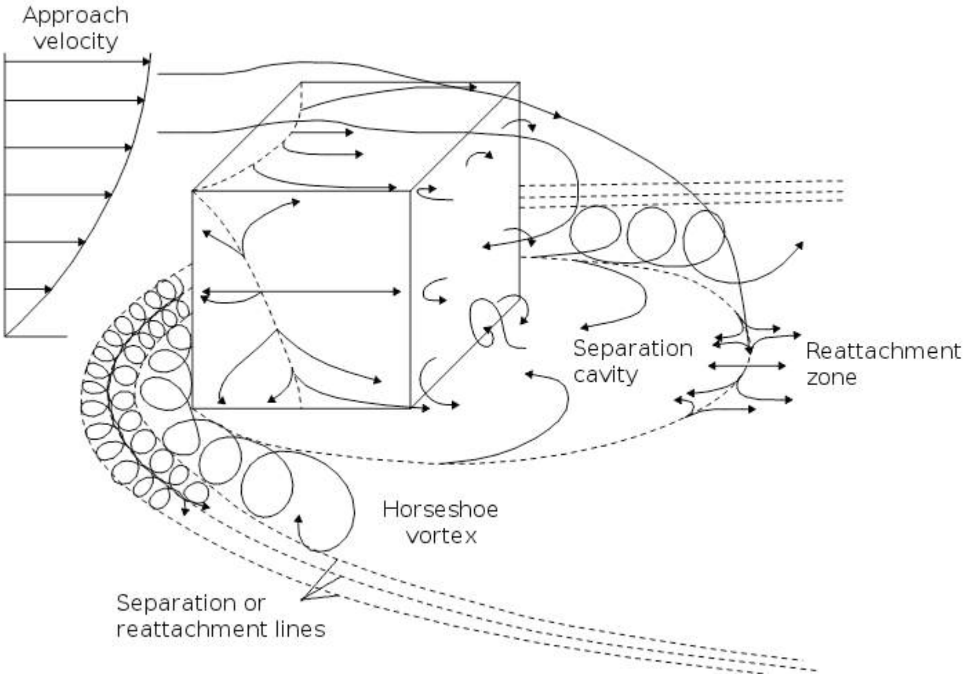

Fig. 1. Mean streamline patterns about a building, after Peterka et al.[16]

3. Methodology

3.1. Description of the campus and surroundings

The De Montfort university (DMU) campus is located near the centre of the city of Leicester in the UK. Fig. 2 shows the area of the campus and surrounding buildings, indicating the average building heights. This case is of particular interest as it has a complex geometry with large variation in building heights. The buildings included in the computational domain are the buildings within the De Montfort university campus and the surrounding buildings to a radius of about 300 m around the campus—in accordance with best practise guidelines [17]. The location and range of buildings found on the campus mean that it is similar to much of the city-centre environment.

The university campus has been modelled in significant detail, while the surrounding buildings are modelled in simpler form with flat roof surfaces. Inside the campus there are a few high-rise buildings. The highest building in the model is located at the edge of the buildings and is 50.1 m high. In the immediate outer area, there is a variation in buildings type, ranging from 2 storey residential buildings to high-rise and medium-rise buildings, but further away the terrain is more homogeneous, including mostly cultivated areas with low vegetation and scattered areas of low residential houses of 1 or 2 storeys, light industrial buildings and clumps of forest. There are very few trees and the underlying terrain is relatively flat. Turbulence is largely driven by the low-rise buildings downstream of the main area of interest.

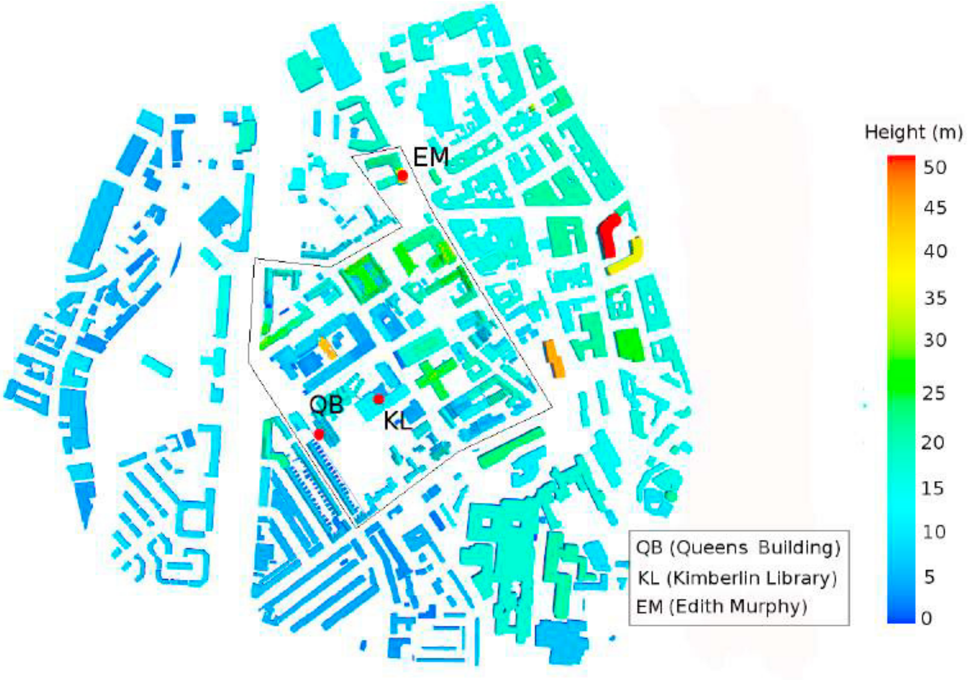

Fig. 2. De Montfort University campus and the surrounding buildings indicating the average building heights.

3.2. Full-scale measurements

Three ‘Windmaster Pro’ 3D ultrasonic anemometers (Gill instruments Ltd) were used for high resolution wind measurements at the university campus during the period January 2015 - March 2016. Data were sampled at 20 Hz and averaged over periods of 1 sec, 1 min and 10 min where the sampled data were complete over the corresponding time intervals. The anemometers were positioned as follows:

- One was installed on the roof of the Edith Murphy (EM) building (Fig. 3(a)), the second highest building on the university campus (30 m tall). The anemometer was mounted on a 5.10 m mast which extends 2.35 m above the uppermost roof level so that the total height for these observations was 36.35 m from the ground. This served as a reference measurement point with the other points deliberately positioned where there was expected to be reduction in speeds and directional effects due to adjacent building features.
- The second anemometer was mounted on top of a 5.10 m mast, positioned on the roof of the Kimberlin library (KL) building (14.60 m tall) (Fig. 3(b)) so that the total measurement height was 21 m from the ground. This position was chosen as it is partly in the shadow of the Queens Building (QB), as opposed to the previous well-exposed EM location.
- The third anemometer was used for short-term (2hours) measurements (SM) using a telescopic mast at the rear of the QB (Fig. 3(c)) and the observation heights range from 9.95 m to 11.35 m from the ground.

Fig. 3. (a) Edith Murphy; (b) Kimberlin Library; (c) Queens Building.

3.3. Computational domain and mesh

The computational domain is firstly defined by CAD data representing 1100×1100 m^2 of the De Montfort university campus and the surrounding buildings. The dimensions of the complete domain (Fig. 4) are (Length)×(Width)×(Height) = 2200×1300×300 m^3. This accommodates an upstream length of 5H (with H being the height of the highest building: 50 m), a downstream subdomain length of 15H and a height of 6H. The geometry is characterized by a maximum blockage ratio of 2% —less than the 3% suggested by Tominaga et al.[18]. The lateral boundaries have been placed 2H from region of interest in accordance with best practice guidelines [18,19].

This case is representative of complex urban environments and it was of practical interest to apply the OpenFOAM `snappyHexMesh` tool to mesh the case as this offers a high degree of automation and good parallel efficiency. The near-wall regions around the buildings were refined sufficiently to match the underlying geometry so that the total number of cells for the whole domain was approximately 24 million.

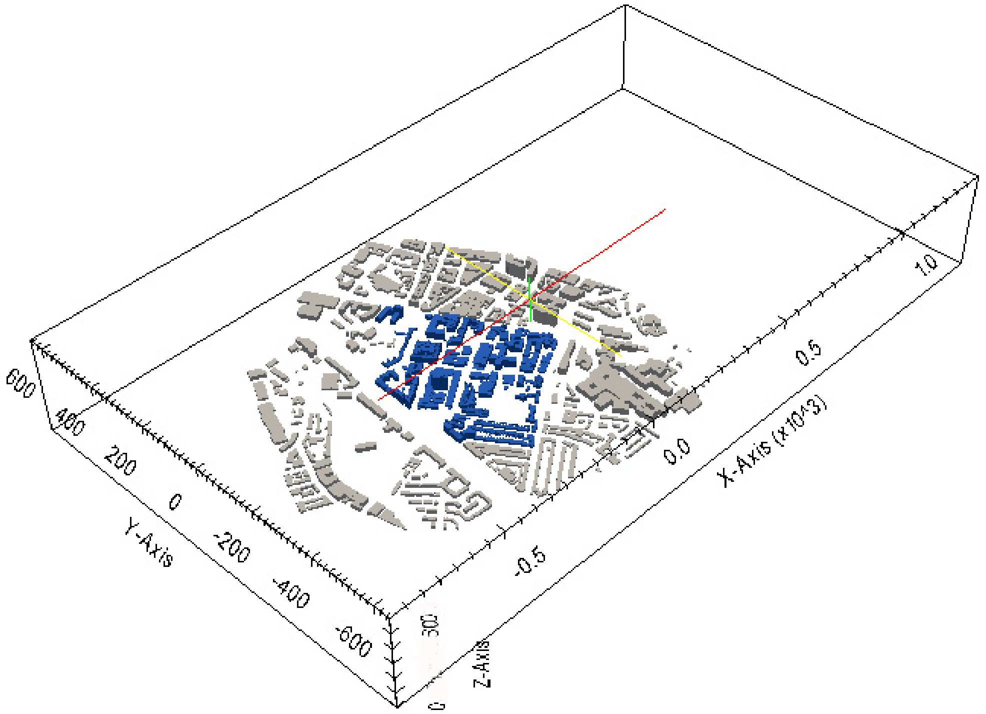

Fig. 4. Computational domain.

3.4. Turbulence model and Boundary conditions

The DES Spalart Allmaras (DES-SA) model was used with the `pimpleFoam` transient solver. The simulation time during which the data were averaged was $T = 2 \cdot T_0$, where T_0 is the time required for a nominal cross-flow through the domain. The initial simulation period corresponded to more than $T = 4.5 \cdot T_0$. The boundary conditions are summarized in Table 1.

Table 1. Boundary conditions.

Boundary	Condition
Inlet	Power law velocity profile, $U(z) = U_s(\frac{z}{z_s})^a$, a=0.24 [20]
Outlet	Zero gradient condition
Lateral and upper surfaces	Symmetry conditions
Building and ground surfaces	Spalding's wall function

4. Results

To validate the CFD model, the measurements at the two fixed positions (EM and KL) and the mobile measurements at QB were compared with the CFD predictions at the corresponding points in the model. The experimental data were sampled at 20 Hz and were averaged over 10 minute periods. To make comparisons, wind speed ratios were calculated by dividing the wind speed values at the locations of the anemometers (U_{KL}, U_{SM1}, U_{SM2}) by the reference wind speed. The Edith Murphy building was used as the reference wind speed ($U_{EM,ref}$) for the Kimberlin Library and the Kimberlin Library was used as the reference wind speed for the short-term measurements (SM) ($U_{KL,ref}$).

The wind speed ratios were also calculated from the CFD simulations using the statistically averaged flows from the DES calculations and Fig. 5 compares them with the measurements. There is a good agreement with a deviation of less than 10% apart from the North direction of SM1 where a discrepancy of 15% was found. In this position, only two hours measurements were carried out and hence there is lower confidence in these data. Also, noticeable speed gradients exist at the SM site (Fig. 6) and hence the results are sensitive to small variations in measurement/calculation position.

The second validation quantity of interest is the difference in wind direction between the reference location and measured by the other anemometers and the corresponding CFD results. These data are compared in Fig. 7 which shows the difference between measured and simulated values for the angle of deviation ($\Delta\phi$) between the KL and the reference wind at EM ($\Delta\phi = \phi - \phi_{ref}$). These differences are less than 3 degrees, indicating good agreement. The results for the short-term measurements (SM) are less good in terms of $\Delta\phi$: 50% to 70% deviation. This might be attributed to the fact that the anemometer was difficult to align accurately to the North direction with the temporary mast used for the short-term measurements.

Summarising, the overall agreement in wind magnitude is acceptable over a range of directions and suggests the approach is robust. Agreement in terms of direction is quite good, and the discrepancies at some points can be attributed to the difficulty to extract the exact coordinates of the points of measurements and the uncertainty in the SM anemometer North direction.

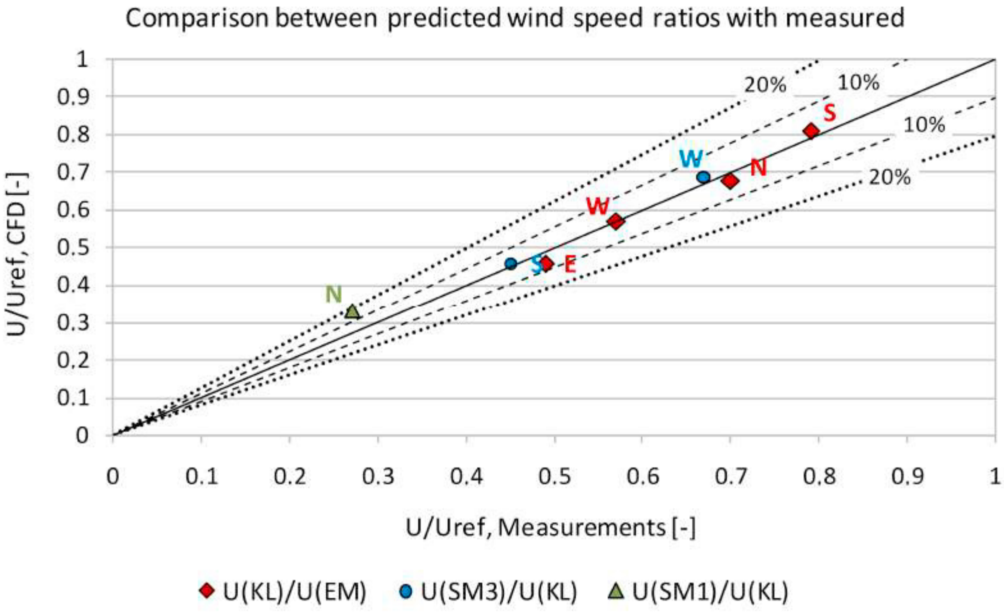

Fig. 5. Comparison between numerical and experimental (10 degree interval) wind speed ratios (U/Uref) in the locations of anemometers for four wind directions -North (N), East (E), South (S), West (W).

5. Discussion

Limited CFD studies of complex urban areas have been validated using field measurements [4,6,7,21] and none of these have used a DES model. In this work, the prediction capabilities of the unsteady CFD approach were examined in the study of wind flows at the DMU campus and the results compared very favourably with high frequency on-site anemometer data. This benefit came at some cost in terms of the computational resources required and time; super computers were engaged to perform the simulations and it took almost a year to obtain the results (during this time a lot more cases were tested before setting-up the final configuration of the models). Nevertheless this approach is less computationally demanding than full LES approaches but produces similar quality solutions for the wake regions of most interest. Consequently, although it is significantly more computationally demanding than RANS calculations,

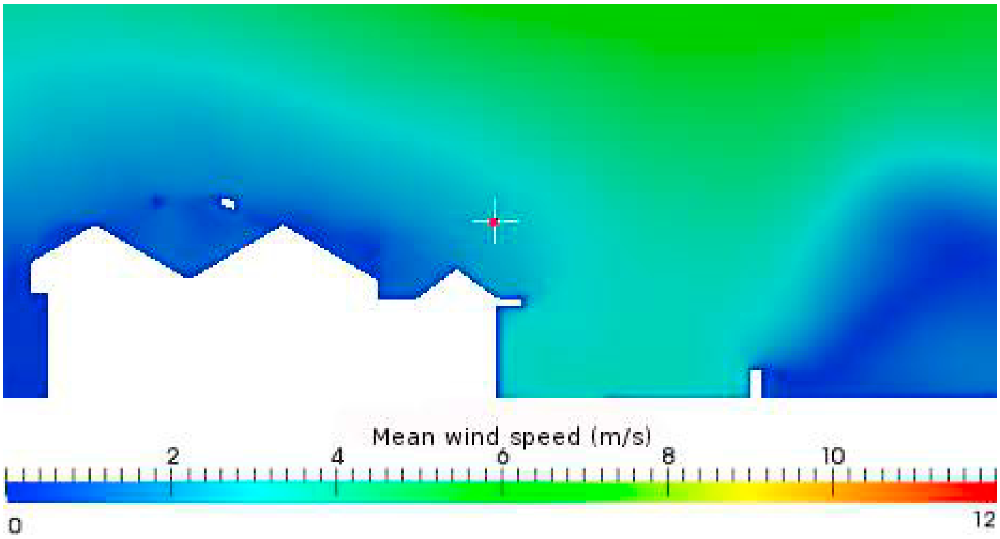

Fig. 6. Wind speed mean in the position (red point) of SM1 for North wind direction at xz plane.

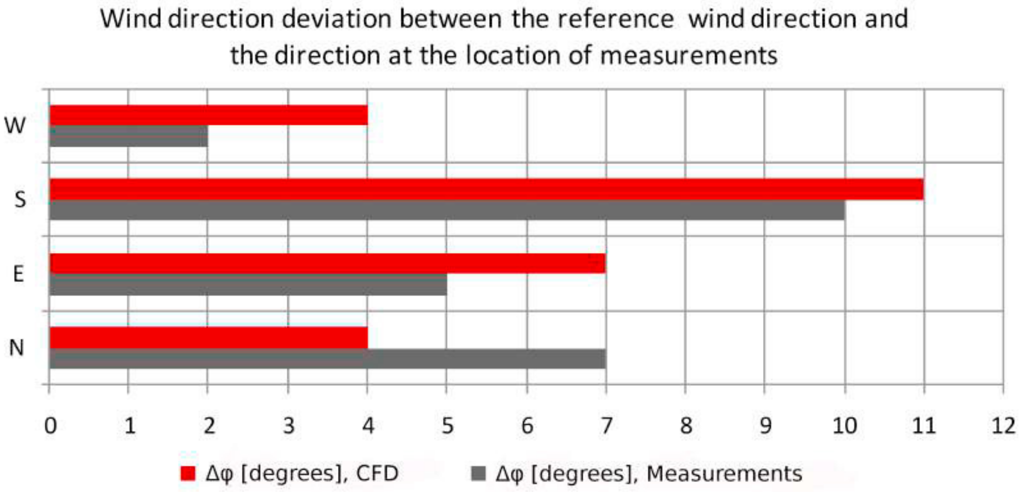

Fig. 7. Comparison between numerical and experimental angle deviation of wind direction between the location of anemometers at KL and the reference wind at EM ($\Delta_\phi = \phi - \phi_{ref}$) for four wind directions -North (N), East (E), South (S), West (W).

offers improved prediction of flows in wake regions and the accuracy required for micro wind turbine applications. [22].

The robustness of DES implementation should be further investigated and more studies validated using field measurements are necessary to test its ability to predict wind flows in complex urban areas. The critical component for this action is the use of ultrasonic anemometers which allow for accurate, high frequency, three-dimensional measurements of flow speed and turbulence properties. As the built environment is highly turbulent and can substantially effect turbine performance, studies of turbulence intensity prediction above roofs are required and rigorous CFD-based evaluation methods need to be developed further.

As regards the number of the wind directions simulated in a wind energy assessment, they should be chosen based on how much the wind direction effects the wind speed ratios. Generally, the smaller the wind direction intervals, the more accurate the results. However, it is impractical to do calculations for each wind direction and the measurements should be clustered in wind direction intervals. Intervals of 45 to 10 degrees i.e. 8 to 36 wind directions are proposed, based on how much the change in wind direction effects the wind speed ratio [7,23]. In this work, the wind flow was calculated for four wind directions for validation purposes. However, if one wants to investigate further the response of this model to the wind direction, wind simulations of smaller wind intervals are required and the impact of the wind direction on wind speed ratio at a wider range of locations should also be investigated.

6. Conclusions

In this paper the predictive capabilities of the DDES-SA model using the Open-FOAM CFD library were investigated. The DDES-SA approach was applied at the DMU campus and the results compared with high frequency anemometer data. The mean velocity predictions above rooftop at a well exposed building and at a partly sheltered building were examined and found to be in good agreement with the anemometer data. Very limited CFD studies of complex urban areas have been validated using field measurements in this way. The results of this study have offered good evidence of the robustness and accuracy of the DES approach, which is well suited to analysis of wind energy potentials in complex urban environments.

Acknowledgements

We would like to acknowledge the industrial collaborator Sarah Brown at Infoterra Ltd, who provided the geometry data for Leicester. Without her involvement this project would not have been able to progress.

References

[1] Drew, D., Barlow, J., Cockerill, T.. Estimating the potential yield of small wind turbines in urban areas: A case study for Greater London, UK. Journal of Wind Engineering and Industrial Aerodynamics 2013;115:104–111.

[2] Millward-Hopkins, J., a.S. Tomlin, , Ma, L., Ingham, D., Pourkashanian, M.. Assessing the potential of urban wind energy in a major UK city using an analytical model. Renewable Energy 2013;60:701–710.

[3] Gagliano, A., Nocera, F., Patania, F., Capizzi, A.. Assessment of micro-wind turbines performance in the urban environments: an aided methodology through geographical information systems. International Journal of Energy and Environmental Engineering 2013;4(1):43.

[4] Yang, A.S., Su, Y.M., Wen, C.Y., Juan, Y.H., Wang, W.S., Cheng, C.H.. Estimation of wind power generation in dense urban area. Applied Energy 2016;171:213–230.

[5] Karthikeya, B., Negi, P.S., Srikanth, N.. Wind resource assessment for urban renewable energy application in singapore. Renewable Energy 2016;87, Part 1:403 – 414.

[6] Tabrizi, A.B., Whale, J., Lyons, T., Urmee, T.. Performance and safety of rooftop wind turbines: Use of CFD to gain insight into inflow conditions. Renewable Energy 2014;67:242–251.

[7] Kalmikov, A., Dupont, G., Dykes, K., Chan, C.. Wind power resource assessment in complex urban environments : MIT campus case-study using CFD Analysis. In: AWEA WINDPOWER Conference. 2010,.

[8] Breuer, M., Jovii, N., Mazaev, K.. Comparison of des, rans and les for the separated flow around a flat plate at high incidence. International Journal for Numerical Methods in Fluids 2003;41(4):357–388.

[9] Franke, J., Hirsch, C., Jensen, A., Krüs, H., Schatzmann, M., Westbury, P., et al. Recommendations on the use of CFD in wind engineering, COST Action C14. European Science Foundation COST Office; 2004.

[10] Cheng, Y., Lien, F., Yee, E., Sinclair, R.. A comparison of large Eddy simulations with a standard kϵ Reynolds-averaged NavierStokes model for the prediction of a fully developed turbulent flow over a matrix of cubes. Journal of Wind Engineering and Industrial Aerodynamics 2003;91(11):1301–1328.

[11] Petry, A.P., Loredo-Souza, A.M., Filho, D.G.R.D.F., Mattuella, J.M.L., Gusberti, F.W.. Evaluation of Experimental Wind Tunnel, RANS and LES Analysis of Wind Flow Over a Complex Terrain. In: The European Wind Energy Association, Paris-France. 2015,.

[12] Rodi, W.. Comparison of les and rans calculations of the flow around bluff bodies. Journal of Wind Engineering and Industrial Aerodynamics 1997;69:55 – 75.

[13] Villiers, E.. The Potential of Large Eddy Simulation for the Modeling of Wall Bounded Flows. Ph.D. thesis; 2006.

[14] Spalart, P.R., Jou, W., Strelets, M., Allmaras, S.R.. Comments on the Feasibility of LES for Wings, and on a Hybrid RANS/LES Approach 1997;.

[15] Salim, S.M., Ong, K.C., Cheah, S.C., Ntroduction, I.I.. Comparison of rans , urans and les in the prediction of airflow and pollutant dispersion. In: World Congress on Engineering and Computer Science 2011 Vol II, San Francisco, USA. 2011,.

[16] Peterka, J., Meroney, R., Kothari, K.. Wind flow patterns about buildings. Journal of Wind Engineering and Industrial Aerodynamics 1985;21(1):21 – 38.

[17] Franke, J., Hellsten, A., Schlunzen, K.H., Carissimo, B.. The cost 732 best practice guideline for cfd simulation of flows in the urban environment: a summary. International Journal of Environment and Pollution 2011;44(1-4):419–427.
[18] Tominaga, Y., Mochida, A., Yoshie, R., Kataoka, H., Nozu, T., Yoshikawa, M., et al. AIJ guidelines for practical applications of CFD to pedestrian wind environment around buildings. Journal of Wind Engineering and Industrial Aerodynamics 2008;96(10):1749 – 1761.
[19] Franke, J., Hellsten, A., Schlünzen, H., Carissimo, B.. Best practice guideline for the CFD simulation of flows in the urban environment, COST Action 732. European Science Foundation COST Office; 2007.
[20] Choi, E.C.C.. Proposal for unified terrain categories exposures 2009;(1).
[21] Simoes, T., Estanqueiro, A.. A new methodology for urban wind resource assessment. Renewable Energy 2016;89:598 – 605.
[22] Dadioti, R.. Numerical Wind Resource Assessment in Urban Environments. Ph.D. thesis; 2017.
[23] Irshad, W.. Wind Resource Assessment : Statistical and Computational Fluid-Dynamic Analysis. Ph.D. thesis; 2012.