Estimation of Wind Energy Over Roof of C Buildings Faculty of Engineering University of Riau

Iwan Kurniawan,[a,*]

[a)] *Departement of Mechanical Engineering, Universitas Riau, Pekanbaru, 28293, Indonesia*

*Corresponding author: iwan.ktm@gmail.com

Paper History

Received: 10-October-2015
Received in revised form: 20-November-2015
Accepted: 23-Nevember-2015

ABSTRACT

Building-integrated urban wind turbines are promising low-cost renewable energy devices. However, the take-up of urban wind turbines in high density suburban environments is still very interesting to investigate by issues such as: a) low wind speeds; b) high turbulence intensity; and c) the perception of potentially high levels of aerodynamic noise generated by the turbines. This paper presents a numerical study was performed to C building Faculty of Engineering University of Riau using type of RANS turbulence k-e models in the open source code Open FOAM, and the results are compared with published in-situ measurements and published wind tunnel tests. Based on the computational results, site for wind turbine installation above these roofs has been assessed. It has been found that turbulence intensity and wind direction for mounting wind turbine site is 1.8H in the corner location above roof C building.

KEY WORDS: *k-e, Open FOAM, RANS, Urban Wind turbine.*

NOMENCLATURE

API	American Petroleum Institute
ΔT	Temperature Difference in and out
F_T	Thermal Expansion
L_A	Anchor Length
ΔL	Expansion
F_P	Pressure Force
F_F	Friction Force
ε_{sd}	Design Compressive Strain
ε_c	Critical Strain

1.0 INTRODUCTION

Building-integrated urban wind turbines is one of the potentially low-cost renewable sources of energy. Despite their potential, Ledo et al. [7] pointed out that the reasons behind the limited installation of micro-wind turbines in urban areas are the low mean wind speeds, high levels of turbulence and relatively high aerodynamic noise levels generated by the turbines. If a turbine is sited in the wrong location on a dwelling roof, it is possible for the power output to diminish to zero for significant periods of time, evenwhen the wind is blowing strongly. Another reason for the cautious integration of micro-wind turbines within urban areas is the negative reputation of urban wind energy due to the erroneous installations of rooftop wind systems as a signal of support for sustainability without adequate consideration of safety, structural building integrity or turbine performance. Thus, numerical modelling of the wind flow above the building roof is important for the design of residential suburban landscapes. It is expected that more and more houses with integrated wind turbines will be built as sustainability becomes an increasing design driver for new houses in the future [5].

Toja-Silva et al [11] and abohela et al [1] present a review of the opportunities and challenges of urban wind energy that stresses the necessity to perform accurate analyses of the flow behaviour on building roofs, in order to get more information about possible positions of wind turbines to take advantage of the accelerating effect of the wind above the building, the adequate kind of turbine and the estimation of the power generation.

Ledo et al. [7] studied wind flow around pitched, pyramidal and flat roofs under three wind directions (0, 45& 90^o) for the purpose of roof mounting wind turbines, they concluded that the power density above the flat roof is greater and more consistent than above the other roof types and they recommended extending the investigation to include other roof shapes. Phillips et al. [10]

investigated the mounting location for a single winddirection for a gabled roof and recommended extending the investigation to include more roof types and more locations with different wind directions. Mertens [8] analysed flow over a flatroof with a view to developing a small wind turbine siting guide lines focussing on the mounting height.

In the present work, local wind flow characteristics above C building in Faculty of Engineering University of Riau is considered, figure 1 Potential power wind investigated by employing Computational Fluid Dynamics (CFD) to installation location of roof mounted wind turbines.

Figure 1: Campus faculty of Engineering University of Riau

2.0 NUMERICAL ANALYSIS

For modelling wind flow over complex terrain, the Reynolds-averaged Navier-Stokes approach (RANS) combined with a k-epsilon k-ε scheme as turbulence model, is the most common approach in wind engineering. The method provides a fair compromise between computational costs and accuracy. Reynolds decomposition is employed to the variables of the governing equations, whereby each variable is divided into a time-averaged part and a fluctuating part, $u = \bar{u} + u'$, resulting in the two following equations:

$$\rho\left(\frac{\partial \bar{u}_i}{\partial t} + \bar{u}_j \frac{\partial \bar{u}_i}{\partial x_j}\right) = -\frac{\partial \bar{P}}{\partial x_i} + \rho \bar{g}_i + \frac{\partial}{\partial x_j}\left[\mu\left(\frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_i}\right)\right] - \frac{\partial}{\partial x_j}\overline{\rho u_i' u_j'} \quad (1)$$

$$\frac{\partial \bar{u}_k}{\partial x_k} = 0 \quad (2)$$

The terms $\overline{u_i' u_j'}$ as the Reynolds stresses and physically represent the additional stresses due to the fluctuating components of the flow. These Reynolds stresses have been modelled according to a 'Boussinesq' approximation, shown in Eq. (3), an analogy of Newton's friction law:

$$\tau_{ij} = -\rho \overline{u_i' u_j'} = \mu_t \left(\frac{\partial u_i}{\partial x_i} + \frac{\partial u_j}{\partial x_i}\right) + \frac{2}{3}\rho k \delta_{ij} \quad (3)$$

where μ_t is the turbulent viscosity and $k = 1/\overline{2u_i' u_j'}$ is the turbulent kinetic energy (TKE). The 'standard' k-epsilon (k-ε) model, based on a two-equation turbulent energy scheme provides reasonable results in approximately neutral atmospheric conditions, and provides an acceptable estimate of the turbulence intensity through the turbulent kinetic energy term.

For example, some best practice guide lines conclude that the k–ε standard model should not be used in simulations for wind engineering problems, and recommend the improved two-equation models or differential stress models [5,12]. However, some other researchers found that the k–ε standard model maybe better, Wang et al [13]found that the k–ε standard model gives rather better performance than the realizable k–ε, Reynolds stress models (RSM) or renormalization group (RNG) k–ε.

2.1. Inflow Wind Profile

The velocity profile at the inlet boundary of the simulation domain must be accurately modelled to provide valid results of wind simulation in the built environment. The roughness of the ground affects the profile of wind velocity, u, and therefore is necessary to be part of velocity profile simulation. The following equation is used to model the wind profile at the inlet of CFD domain[:

$$u = \frac{u^*}{\kappa}\ln\left(\frac{z + z_0}{z_0}\right) \quad (4)$$

$$k = \frac{u^{*2}}{\sqrt{C\mu}} \quad (5)$$

$$\varepsilon = \frac{u^{*3}}{\kappa(z + z_0)} \quad (6)$$

where u is the inlet velocity (horizontal, of axis x) at height z, u^* is the friction velocity, κ is the von Karman constant, z_0 is the aerodynamics roughness length and C_μ is the turbulence model constant. The comparison between thewind profile and the analytic inlet values is given in figure. 1.

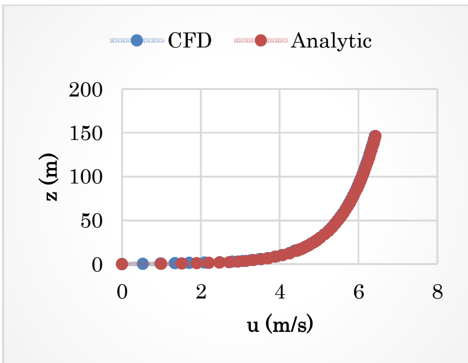


Figure 2: Inlet velocity comparison between analytic and CFD simulation in thevertical section

2.2. Description of the case study and simulation details

Structured hexahedral mesh, shown in Fig. 2. A mesh independence study was carried out to determine the dependence of the flow field on the refinement of the mesh. Final mesh statistics in the present models were: 704730 elements for case 0 degree, case 90 degree and for case 180 degree.

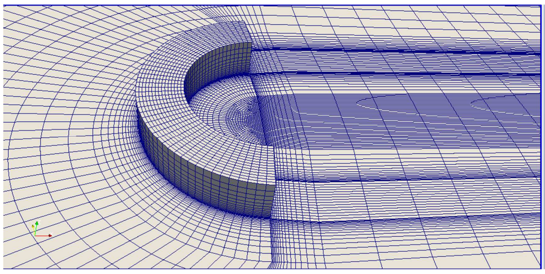
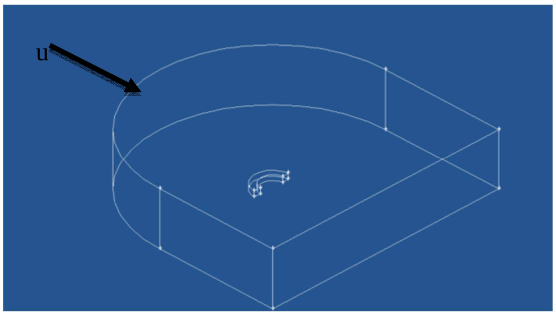
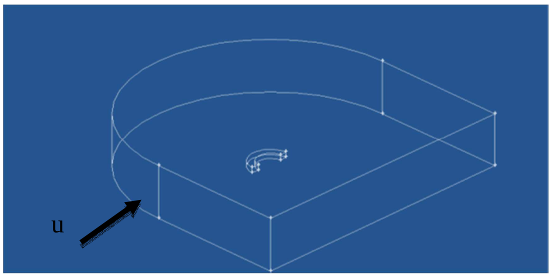


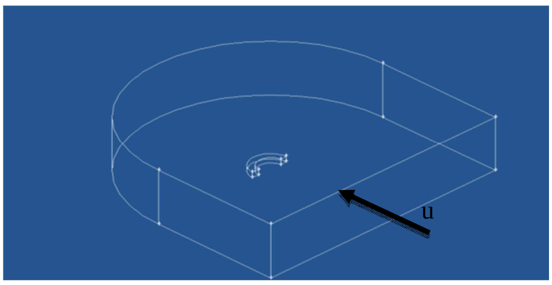
Figure 3: Type and density mesh used in computational



(A)



(B)



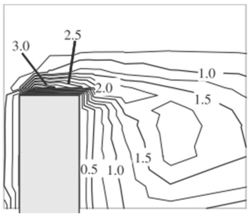
(C)

Figure 4: Computational domain with each orientations (A) 0 degree, (B) 90 degree and (C) 180 degree

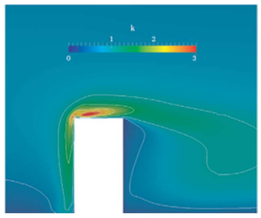
The building models with 15 m height, 15.6 m wide and outer radius 45 m are placed in a rectangular domain for back side models and C shape for front models. Dimensions computational domain are X xY x Z = 1000 x 150 x 1000 m^3.The inlet velocity profile was specified by eq.4-6 (Blocken et al) The downstream boundary was specified as opening with zero relative pressure. The side faces were setas symmetric boundaryand top of the domain were set as wall slip condition. All solid boundaries bottom and C building were set as no-slip walls. Standard turbulent k-ε (TKE) used in computational.

3.0 RESULT AND DISCUSSION

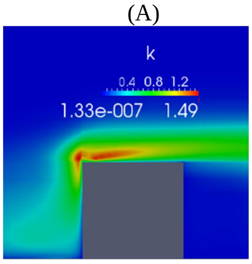
CFD simulation, as a wind assessment tool, is embedded with errors and uncertainties. Hu [4] attributed this to the many physical and numerical variables which might puzzle even experienced users. Thus, Blocken et al. [3] asserted the importance of validating CFD simulations against other wind assessment tools. In this section wind flow around a surface C building in a turbulent channel flow is investigated using the in house CFD code Open FOAM. For validation purposes the results will be compared to published in-situ measurements and published wind tunnel tests.



(A)



(B)



(C)

Figure 5: Comparison of the turbulent kinetic energy (m^2/s^2) at vertical section at the centre of the domain. (A) Durbin-Standard, (B) k-ε standard case A, (C) Exp. Tominagaet all (2008). (note : comparison only trend profile k not value).(Toja et all:11)

3.1 Wind Flow above Building C

Wind flow above the roofs is complex and cannot be predicted from the wind data because of the proximity of the roofs in densely populated suburban residential houses, the flow is highly turbulent and the wind velocity field isvery different from the free stream velocity due to the bluff body effects of the buildings and the evolution of separated regions. It is therefore important that local wind characteristics such as the flow pattern, turbulence intensity and wind velocity need to be carefully analyzed when micro-turbines are to be integrated within these built environments.

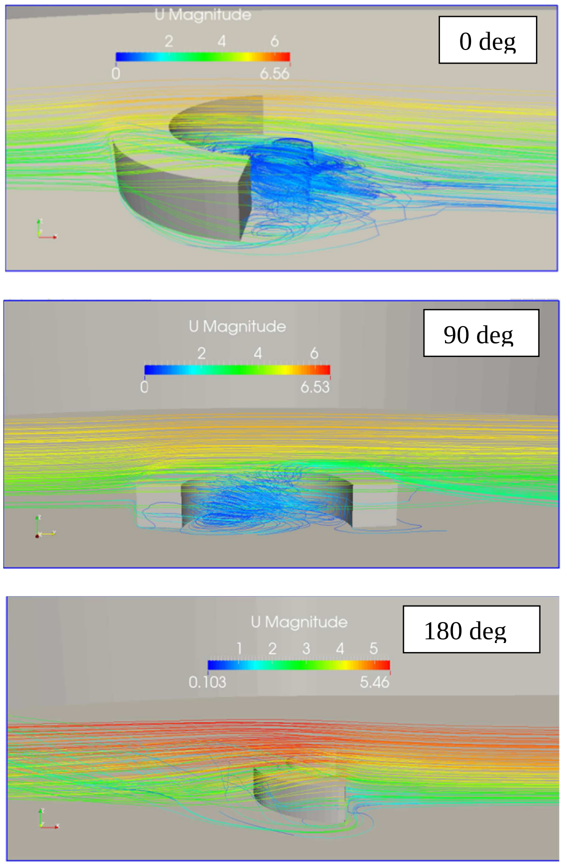


Figure 6: Flow pattern u velocity for three orientation wind inflow

3.2 Flow pattern

In this section the flow patterns above the C building of the threedifferent inflow Fig. 6 showsfor 0 degree, 90 degree and 180 degree. According to figure 6, notanywhere above the Cbulding roof is suitable sitefor installing a turbine, onlythe corner position is recommended as a mounting location.

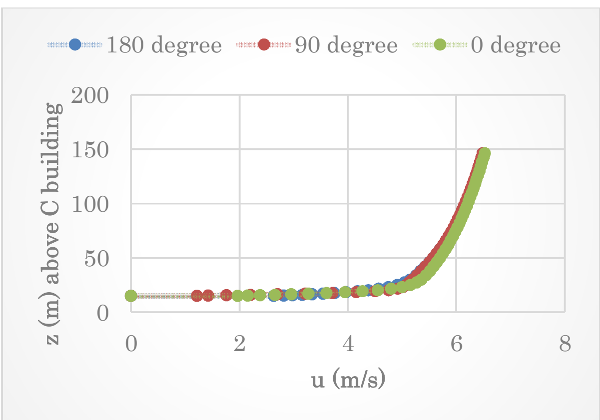


Figure 7: Velocity for different inflow to corner location

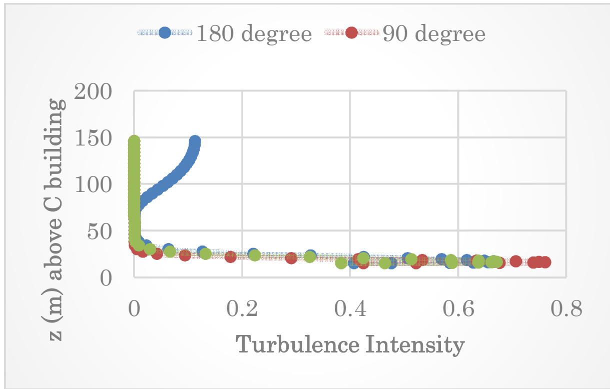


Figure 8: Turbulence Intensity for three different inflow to corner location

Turbulence intensity affects the operability and the lifetime ofwind turbines. Ledoet all, a turbine should not beexposed to wind with turbulence intensity greater than 16-18%.Therefore, it is important to estimate the turbulence intensity atany prospective turbine mounting location. In this study three winddirections, a 0 degree, 90 degree and 180 degree were considered. The results for the 0, 90, 180 degree for z above roof C building, turbulence intensity level decrease with height. Figure 8, turbulence intensity level for < 18 % at 1.8H. In figure 7 & 8, Data turbulence intensity and velocity are mean value at three location to corner location above C building. Three location are x, y and z value shows for table 1 :

Table 1: Three location for x, y and z value data for u and turbulence intensity.

0 degre			90 degree			180 degree			point
x	y	z	x	y	z	x	y	z	
-10	-30	15	-10	-30	15	-10	-30	15	point 1
-10	-30	146	-10	-30	146	-10	-30	146	Point 2
-20	-30	15	-10	-35	15	-20	-30	15	point 1
-20	-30	146	-10	-35	146	-20	-30	146	Point 2
-30	-30	15	-10	-40	15	-30	-30	15	point 1
-30	-30	146	-10	-40	146	-30	-30	146	Point 2

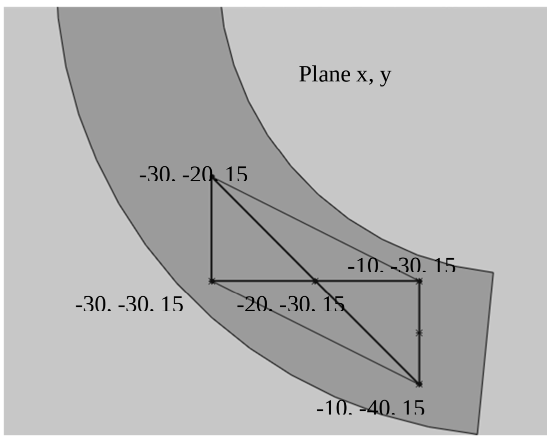


Figure 9: Estimation location mounting urban wind turbines on the roof C building

3.3 Wind velocity result

In the previous section, favourable mounting location for windturbine has been identified based on the level of turbulenceintensity are 1.7H.

For wind direction, it is observed an increase of wind velocity in all inflow orientation. Wind power density (Eq. (7& 8)) is a useful way to evaluate maximum thewind power available at a potential site.

$$\text{Power Density} = 0.5\rho u^3 \quad (7)$$

$$\%\, power\ density = \frac{u - u_{ref}}{u_{ef}} x100 \quad (8)$$

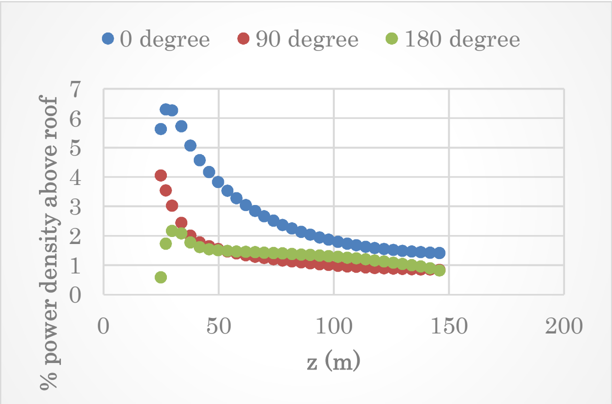


Figure 10: potential power density as function of wind direction

It can be used to compare the power available above the three wind direction above roof C building. The dependence of the power density on the winddirection for a turbine located at the corner of the c building. Maximum increase power density is 1.8H, a location for mounting wind turbine is shown in Fig. 9 and 10. In all cases, the wind turbine hub is 27 m above c building at corner location (tabel 1).

4.0 CONCLUSION

In this investigation, CFD simulations of the wind flow around asingle building were performed within house code openFOAM using k-ε RANS turbulence models, and the results were compared with published in-situ measurements and published wind tunnel tests.

Wind flow simulations above C building with wind direction have been performed. The simulations looked into the wind flow characteristics in terms of turbulence intensity, wind velocity and wind flow pattern. Based on the computational results, site for wind turbine installation above these roofs has been assessed. It has been found that turbulence intensity and wind direction for mounting wind turbine site for 1.8Hin the corner location above roof C building according to table 1 data. Further study of the wind flow characteristics with the turbinemounted on the roof will also need to be carried out. In this case,modelling using LES might be used to study the unsteady wind flowpattern and to evaluate the feasibility of a roof turbine installationbased on the annual wind power density. Feasibility urban wind turbines farm need to investigate above C building.

ACKNOWLEDGEMENTS

The authors would like to convey a great appreciation to lembagaPenelitiandanPengabdian (LPPM-UR) university Of Riau for supporting this research.

REFERENCE

1. Abohela, I., Hamza, N., Dudek, S., (2013). Effect of roof shape, wind direction, building height and urban configuration on the energy yield and positioning of roof mounted wind turbines. Renew. Energy vol.50, pp.1106–1118.
2. Blocken B, Stathopoulos T, Carmeliet J, (2007). CFD simulation of the atmospheric boundary layer: wall function problems. Atmospheric Environment Vol. 41 pp. 238-52
3. Blocken B, Stathopoulos T, Carmeliet J, Hensen JLM,(2011). Application of computational fluid dynamics in building performance simulation for the outdoor environment: an overview. Journal of Building Performance Simulation. Vol4 pp. 157-84.
4. Hu C-H. (2003). Proposed guidelines of using CFD and the validity of the CFD models in the numerical simulations of wind environments around buildings. In: School of the built environment. Edinburgh: Heriot-Watt University;. p. 177.
5. J. Franke, C. Hirsch, A.G. Jensen, H.W. Krüs, M. Schatzmann, P.S. Westbury, S.D.Miles, J.A. Wisse, N.G. Wright, (2004). Recommendations on the use of CFD in wind engineering, in: J.P.A.J. van Beeck (Ed.), Proc. Int. Conf. Urban Wind Engineering and Building Aerodynamics. COST Action C14, Impact of Wind and Storm on City Life Built Environment, 5–7 May, von Karman Institute, Sint-Genesius-Rode, Belgium,.
6. J. Hang, Z.W. Luo, M. Sandberg, J. Gong, (2013). Natural ventilation assessment in typical open and semi-open urban environments under various wind directions, Build.Environ. Vol.70 pp.318–333.
7. Ledo, L., Kosasih, P.B., Cooper, P., (2011). Roof mounting site analysis for micro-wind turbines. Renew. Energy vol.36, pp 1379–1391.
8. Mertens S. (2003). The energy yield of roof mounted wind turbines. Wind Engineering. vol. 27, pp.507-18.
9. Peacock AD, Jenkins D, Ahadzi M, Berry A, Turan S. (2008). Micro wind turbines in the UK domestic sector. Wind Energy and Buildings, vol. 40,pp.1324-33.
10. Phillips R.(2007) Micro-wind turbines in urban environments: an assessment. Bracknell: IHS BRE Press for BRE Trust.
11. Toja-Silva, F.,Carlos Peralta, Oscar Lopez-Garcia, Jorge Navarro.(2015). Roof region dependent wind potential assessment with different RANS turbulence models.J. of wind Eng. And Ind. Aerodyn.vol.142, pp. 258-271.
12. Tominaga, Y., Mochida, A., Murakami, S., Sawaki, S., (2008). Comparison of various revised it k–є models and LES applied to flow around a high-rise building model with 1:1:2

shape placed within the surface boundary layer. J. Wind Eng.Ind. Aerodyn. Vol,96. pp 389–411.
13. Wang, B, L.D. Cot, L Adolphe, S. Geoffroy, J. Morchain. (2015). Estimation Of Wind Energy Over Roof Of Two Perpendicular Buildings. Energy an d Building. Vol. 88, pp 55-67.
14. Y. Tominaga, A. Mochida, R. Yoshie, H. Kataoka, T. Nozu, M. Yoshikawa, T. Shirasawa, (2008). AIJ guidelines for practical applications of CFD to pedestrian wind environment around buildings, J. Wind Eng. Ind. Aerodyn. 96 (10–11) 1749–1761.