Journal of Cleaner Production 167 (2017) 306–316

Contents lists available at ScienceDirect

Journal of Cleaner Production

journal homepage: www.elsevier.com/locate/jclepro

Harvesting wind energy in low-rise residential buildings: Design and optimization of building forms

Hong Zhou [a], Yujie Lu [b,*], Xiaodong Liu [c], Ruidong Chang [b], Bo Wang [c]

[a] Department of Civil Engineering, School of Architecture and Civil Engineering, Xiamen University, Xiamen, Fujian, 361005, China
[b] Department of Building, School of Design and Environment, National University of Singapore, 117566, Singapore
[c] Department of Architecture, School of Architecture and Civil Engineering, Xiamen University, Xiamen, Fujian, 361005, China

ARTICLE INFO

Article history:
Received 30 March 2017
Received in revised form
21 June 2017
Accepted 19 August 2017
Available online 24 August 2017

Keywords:
CFD
Low-rise building
Renewable energy
Wind power
Micro-wind
Wind turbine

ABSTRACT

As a renewable energy source, wind power has received growing attentions, but mostly been utilized in wind power farms or high-rise buildings where the wind with high speed is available. Few studies focus on the micro-wind utilization in low-rise buildings due to the bottleneck that the wind speed cannot meet the minimal speed requirement of small wind turbines. This paper aims to identify the optimal building design which could enable the harvesting of the maximal micro-wind power around low-rise residential buildings. Based on the comparison among different building shapes and the computational fluid dynamics (CFD) analysis via software *Phoenics*, this paper identified that the building shape of "composite prism" could enable the harvesting of the most micro-wind power. The identified building shape was then tested in a simulated environment of a residential community in Pingtan Island, China. The local wind conditions, long-term community planning, and the requirement of comfort level were all considered in the simulation model. The result shows that the potential of utilizing wind energy in low-rise residential buildings is huge by adopting the proposed building shape of "composite prism". The finding has significant implications for renewable energy utilization in built environment.

1. Introduction

Buildings consume a significant portion of energy worldwide. For instance, building lifecycle energy consumption accounts for 46.7% of the total energy consumption in China (Chang et al., 2015). In the US, around 40% of the energy and three-quarters of electricity every year is consumed by buildings (Han et al., 2015; Srinivasan et al., 2012). Since the global energy crisis is becoming aggravated, the utilization of renewable energy becomes priorities for many industries in various countries (Chang et al., 2017). With minimal impacts on the environment, wind energy has received growing attentions from the construction and building industry. For instance, regarding the industry development of utilizing wind turbines in buildings, Bahrain World Trade Center became the first building with large-scale wind turbines in the world in 2008. In this building, there are three sets of turbines, each of which has a diameter of 29 m (Reuters, 2007). Another example is the Miami Cor Tower, which has multiple horizontal wind turbine installed on four exterior walls at the top of the building (Archdaily, 2010). Situated in Guangzhou, China, the Pearl River Tower completed in 2010 makes full use of wind power to not only generate electricity, but also reduces the compound force of the wind on the building (SOM, 2010).

Various studies have investigated the use of wind power generation technology in buildings, with a focus on multi-layer industrial buildings and super high-rise or high-rise buildings (Heo et al., 2016; Millward-Hopkins et al., 2013; Wang et al., 2015). With the increasingly higher computing capacity of modern computers, computational fluid dynamics (CFD) theory has become an important tool to numerically simulate the wind flow around buildings (Heo et al., 2016; Ledo et al., 2011). An emerging field of study is using CFD to identify the optimal location or building forms which could accelerate wind speed for wind turbines, namely building augmented wind turbine (BAWT). For instance, CFD simulation was adopted to study the effect of different building forms on wind energy capture, thereby evaluating the concentration effect of different building shapes (Abe and Ohya, 2004; Mertens, 2002; Phillips et al., 2002).

There are generally two common positions for mounting wind

* Corresponding author.
E-mail address: luy@nus.edu.sg (Y. Lu).

http://dx.doi.org/10.1016/j.jclepro.2017.08.166
0959-6526/

turbines in buildings, namely the rooftop of buildings, and the space between two buildings (Abe and Ohya, 2004). Commonly adopted by high-rise buildings, the placement of wind turbines on the building's rooftop aims to capture the high-velocity wind on great height, which has received many attentions from scholars (Abohela et al., 2013; Blanch, 2002; Lu and Sun, 2014). Ledo et al. (2011), for instance, adopted CFD to study the wind flow characteristics in different roof profiles, namely pitched roofs, pyramidal roofs and flat roofs, thereby finding the optimum turbine mounting locations on the roof (Ledo et al., 2011). Similarly, Toja-Silva et al., 2016 adopted CFD simulation to identify the optimal building roof shape for utilizing wind energy, revealing that the "Slender shapes" are the most effective building shapes for wind energy exploitation, leading to a higher wind velocity and a lower turbulence intensity (Toja-Silva et al., 2016). In another study, the feasibility of wind power utilization in high-rise buildings of Hong Kong was explored through CFD simulations (Lu and Sun, 2014). This study concluded that wind power utilization in high-rise buildings is very promising theoretically because of the heights of high-rise buildings and the concentration effect of buildings.

Due to the high speed of wind flow around high-rise buildings, studies on placing wind turbines between buildings also focus on high-rise buildings, with low-rise buildings almost ignored by previous studies (Ledo et al., 2011; Li et al., 2016; Lu and Ip, 2009). In this position, the concentration-effect induced by the channel between two buildings is used to accelerate the wind for power generation. For instance, Ayhan and Sağlam (2012) reviewed the development of building-mounted wind power systems and used CFD to simulate the wind flow between buildings with different building distances (Ayhan and Sağlam, 2012). However, they only studied the wind flows around high-rise building without considering medium- and low-rise buildings. Similarly, Heo et al. (2016) studied the aerodynamic power output of an 110 kW wind turbine by CFD simulation, and discovered that due to the concentration effect caused by buildings, the power output of 110 kW BAWT is higher than that of an 110 kW stand-alone wind turbine (Heo et al., 2016). However, the studied 110 kW wind turbine has a rotor diameter of 22.7 m, which determines that it could not be utilized in low-rise buildings.

To summarize, most of the current industrial practices and BAWT studies have focused on utilizing wind power in high-rise commercial buildings. There is a lack of research on using micro-wind in low-rise residential buildings due to a major challenge: the wind flow around low-rise residential buildings is generally deemed to have insufficient speed to drive the wind turbines. However, with the recent development of small-scale portable silent turbines, the application of micro-wind turbines in low-rise residential buildings shows great potentials. For instance, Dabiri (2011) forecasted that micro-wind turbines could generate cheaper wind power than large wind turbines. To utilize wind power to its fullest potential and also enrich the existing BAWT research field, the use of micro-wind turbines in low-rise residential buildings needs to be explored. The key issue is how the wind flows around low-rise residential buildings could be accelerated, thereby enabling the use of micro-wind turbine and improve its efficiency.

To respond to this major challenge, this paper adopts CFD to explore the optimal building design that could accelerate the speed of wind flow for micro-wind turbines. Due to the low height of low-rise residential buildings, this paper mainly explores the amplification effect on wind speed caused by the channel between two buildings through placing wind turbines between buildings, as shown in Fig. 1. The simulation is set in the context of China which is estimated to grow significantly in wind capacity from 1300 GW to 2300 GW, with the annual wind output from 2000 TWh reaching to 3500 TWh (He and Kammen, 2014). With a vast territory and the huge population, China is suitable to utilize renewable energy in buildings (Chang et al., 2016; Zhao et al., 2016a,b). China's *Twelfth Five-year National Science and Technology Plan* attaches great importance to the utilization of wind power in residential buildings (National Energy Administration, 2012). Based on an empirical case of residential developments in China, this study adopts CFD to simulate the wind flows around low-rise buildings with different geometries to identify the optimal building form.

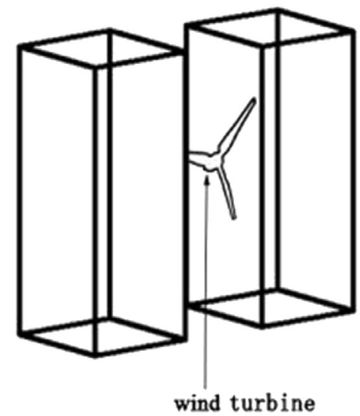

Fig. 1. The analyzed placement of wind turbines in this study.

2. Methodology

2.1. CFD simulation method

CFD is an important method to analyse building ventilation and air flow. Being the most validated and widely applied approach, the standard $k-\varepsilon$ turbulence model is adopted in this study. Fluid dynamics flow control equation is a differential equation that consists of continuity equation, momentum equation and energy equation.

Continuity equation is used to describe the mass conservation and its attributes during the flow process of fluid. The continuity equation of incompressible fluid can be expressed as:

$$\nabla \cdot \vec{w} \tag{1}$$

where, $\vec{w}$ is relative velocity; ∇ is Hamilton operator and can be calculated by. $\nabla = \frac{\partial}{\partial x}\vec{i} + \frac{\partial}{\partial y}\vec{j} + \frac{\partial}{\partial z}\vec{k}$

Equation (1) can be expressed as a rectangular coordinate system, as shown below:

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z} = 0 \tag{2}$$

where, u, v, w are the velocity component of relative velocity $\vec{v}$ on the respective x, y, z coordinate axis in rectangular coordinate system.

Momentum equation is based on Newton second law, which reflects the momentum conservation character in the flow process of fluid. The momentum equation of incompressible fluid can be expressed as:

$$\frac{d(\rho\vec{w})}{dt} = \frac{\partial(\rho\vec{w})}{\partial t} + \rho\vec{w}\cdot\nabla\vec{w} = \rho\vec{F} - \nabla p + \mu\nabla^2\vec{w} - \rho\left[2\vec{\omega}\times\vec{w} + \vec{\omega}\times\left(\vec{\omega}\times\vec{R}\right)\right] \tag{3}$$