Title:
Channel Geometry Optimization for Vertical Axis Wind Turbines in Skyscrapers

Authors:
Seifeddine Kefi kefi.seif@gmail.com, Dept of IELM, HKUST
Ajay Joneja, joneja@ust.hk, Dept of IELM, HKUST
Tim KT Tse, timkttse@ust.hk, Dept of Civil Engg, HKUST
Sunwei Li, Dept of Civil Engg, HKUST

Keywords:
Sustainable Architecture, Vertical Axis Wind Turbines, Green Buildings

DOI: 10.14733/cadconfP.2017.95-99

Introduction:
The desire for introduction of clean energy in urban environments has led to some interest in the use of wind power integrated into modern cities; one way to do so is to integrate wind turbines into skyscrapers, as is seen in some recent architectural projects such as the Pearl River Tower in Guangzhou (Figure 1). Obviously we would like to maximize the power out from such a design, which is a function not only of the parameters of the Vertical Axis Wind Turbines (VAWT), but also on the patterns of prevailing winds in the location of the building, the topography of the surrounding environment, and the shape of the channel in the building where it is installed. In this paper, we shall explore how to optimize the shape of the channel when the other controlling factors have been determined. We focus on VAWT's since they are generally better suited to urban environments due to ease of maintenance and lower noise [9]. The approximate power generated by a VAWT's is given by $P_M = \frac{1}{2}\rho C_p U^3 \frac{\pi d^2}{4}$, where ρ is the air density, d is the turbine diameter, U is the wind velocity, and C_p is the capacity factor, which is the ratio of the power generated to the available power, and is governed by the Betz limit to approximately 59% for Newtonian fluids. While VAWT's may be installed in a few different configurations, we will limit this work to in-building duct installations, such as the one used in the Pearl River Tower in Guangzhou as shown in Fig. 1.

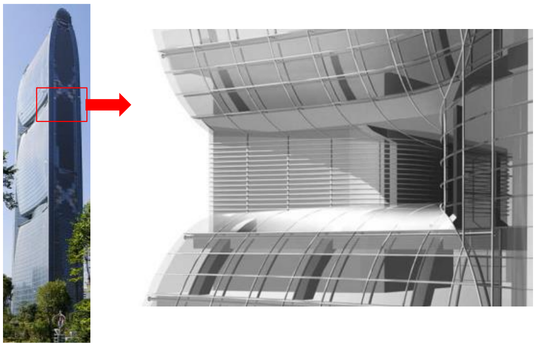

Fig. 1: Pearl River Tower, Guanzhou; SOM architects [12].

Proceedings of CAD'17, Okayama, Japan, August 10-12, 2017, 95-99

Background:

Wind Flow Modeling
Modeling for flow in urban environments requires the use of computational fluid dynamic models. In particular, since the air flow in the atmospheric boundary layer (ABL) is turbulent, modified forms of the standard Navier-Stokes equations are required. One commonly used model is the RANS (Reynolds Average Navier Stokes) model [10]. Since the Reynolds model is under-constrained, Kolmogorov [5] suggested modeling the turbulence using two numbers: the turbulent kinetic energy *k* and the turbulence dissipation rate ε, using which, Hanjalic and Launder [3] developed the k-ε RNG model which has been used successfully in modeling of urban flows and is the model adopted by us in our research.

Solving Wind Flow Models
Since the models that accurately reflect the wind flow are analytically intractable, computational fluid dynamics (CFD) models are solved mainly by numerical techniques. Regardless to the technique used, a successful CFD simulation of the wind in the urban environment depends on three critical points: correct boundary conditions, accurate wind mean profiles and a good mesh quality; since a coarse mesh results in poor output and a very fine mesh will take a long time to converge.
Some early work on wind flow around buildings can be found in [9] and [4]. A more comprehensive study on flow around sets of adjacent tall buildings, possibly with different roof inclinations, was made by Lu et. al in [7]. From the various scenarios studied in that research, some of the key conclusions are: (a) for adjacent tall buildings with a narrow gap and a prevailing wind direction along the gap, there is a significant increase in the wind velocity through the gap; (b) higher turbulence (which is detrimental to efficiency of VAWT's) is observed in the layer up to 3m around building edges; (c) for roof mounted turbines, the tallest building in a group of flat-roofed buildings is the ideal choice; however, if the roof-tops are inclined, then the actual configuration of the buildings dictates the ideal choice.
Our interest is in investigating the optimum shape of a channel housing the VAWT. One study aiming at measuring the performance of the wind turbines placed inside tunnels was conducted by Li et al. [6]. They mainly conducted a series of wind tunnel experiment simulating the Pearl River tower energy performance according to the local climate data, under different conditions (with or without the surrounding buildings, and with or without the presence of the VAWT in the channel). The main findings were that the power generated was affected by the building orientation, and that a contraction in the shape of the channel resulted in increased wind speeds (and therefore higher power). The latter result corresponds to the knowledge used in design of a traditional venturi. Nevertheless, it has been shown (for example, see [1]), that a venturi (or channel) shape with the highest contraction ratio is not necessarily the best in terms of power output. In all cases of such investigations, wind tunnel experiments are necessary for validation of the CFD models.

Wind Tunnel Experiments:
For this study, we use a building that has a square base 30mx30m and a height of 120m. The opening is a circular venturi-shaped tunnel. For the wind tunnel experiment, we use a 1:150 reduced-scale model. The outer circle has a radius of 4.56 cm and the radius at the contraction is 4 cm. The calibrated wind profile presents a wind speed at building height (z=0.8m) of UH=10.171 m/s. We dressed the power law for the wind speed profile to be implemented at the inlet boundary with $\alpha = 0.11$. Other test settings follow Tominaga et al. [11]. The wind velocity and the turbulence intensity, we calibrate the wind profile inspired by the Hong Kong wind code [2]. Figure 2 shows the setup in the wind tunnel. The test conditions ensured that the Von Kàrmàn criterion was matched. The wind speed was measured at five different locations, and our experimental conditions had a Reynolds number = $Re = \frac{UD}{\nu} \approx 1.8\mathrm{x}10^5$ which is in the acceptable range of flows around buildings ($\sim 10^5$).

CFD Simulation:
We use an open source CFD solver, OpenFoam, for our simulations. The first step is to generate the mesh. We developed a simple python script to generate the definition for the mesh in a BlockMeshDict file, which is used by the BlockMesh utility in OpenFoam to create a non-uniform graded mesh. The cells in the region of the interest are small, while those further away from the building are larger. Figure 3 shows the top and elevation views of the mesh.

Proceedings of CAD'17, Okayama, Japan, August 10-12, 2017, 95-99

Fig. 2: (a) The physical scaled model (b) Roughness elements used in wind profile calibration.

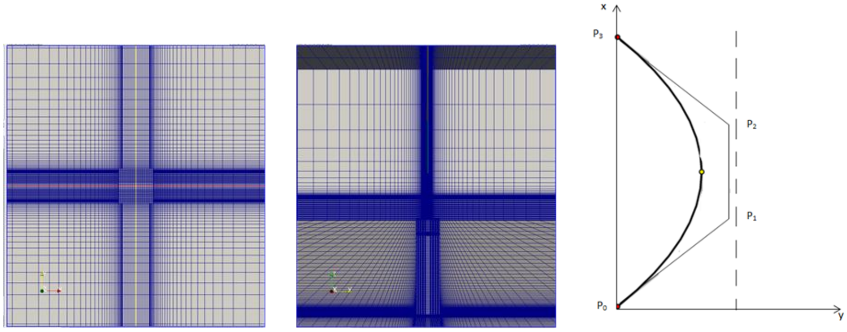

Fig. 3: (a) Top view and (b) elevation view of the simulation mesh in OpenFoam (c) Profile of the test channel shape.

As discussed before, we first create one mesh with a circular profile venturi shape hole in the building to validate the parameters of the simulation with the wind tunnel experimental data. Next, we create a series of test shapes, where the channel is a rectangular profile hole whose height remains constant, but the width varies as per a degree-3 Bezier curve. The control points P_1 and P_2 are moved to change the profile shape. The Semi-Implicit Method for Pressure Linked Equations (SIMPLE) algorithm was used for solving the model. The CFD model results, compared to the wind tunnel data, have an error of less than 5% at the critical positions.

DoE Approach for Optimal Design of VAWT Channels:

The shape of the channel in our study was varied by moving the coordinates (x_1, y_1), (x_2, y_2) of the control points P_1 and P_2. If we use three different values for each of these four variables, then we potentially have to run $3^4 = 81$ experiments. Since each simulation takes several hours to run, we decided to run a partial factorial experiment with 27 runs. Figure 4 shows the flow streamlines though two shapes of the channels (runs #3 and #24).

Finally, we use a statistical interpolation technique, Kriging (see [8] for the theory), to search for the optimum configuration that interpolates the discrete data set generated from the 27 runs. For brevity, we omit the details of the model (which will be presented in the journal form of this paper). Kriging is a stochastic model, where the interpolated values are not deterministic coordinates in the domain, rather, they model a Gaussian process. Under fairly reasonable assumptions on the mean and covariance, the model reduces to a least-squares regression form that can be solved using standard tools in machine learning packages (we use a standard library in R).

Proceedings of CAD'17, Okayama, Japan, August 10-12, 2017, 95-99

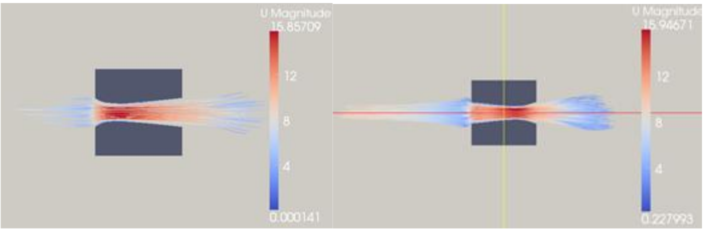

Fig. 4: Horizontal cross-section at the middle of the channel in (a) run #3, and (b) run #24.

We tested our solution by another standard technique, cross-validation. This model fits the optimal model into the results obtained by the partial factorial experiments, with an optimum value at x = (x_1, x_2, y_1, y_2) = (0.0344, 0.1511, 0.0167, 0.0167) (shown in Figure 5).

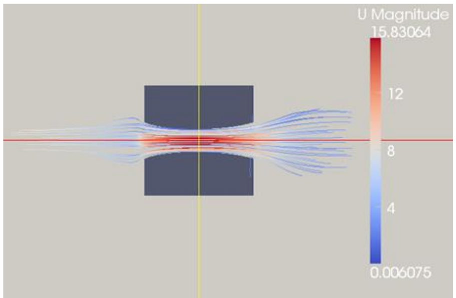

Fig. 5: Streamlines in the optimal configuration given by the Kriging model.

Concluding remarks:

We conclude the paper by summarizing our main contribution, which is the development of a systematic approach using a combination of parametric design, Design of experiments and machine learning tools. The approach is verified by testing on an example building with a simple rectangular channel whose vertical walls are bounded by parametric surfaces and which can house a VAWT. We show that the design does have significant impact on the maximum wind velocity at the contraction point, and therefore on the VAWT output. Our experimental study indicates that channel contraction is a significant factor in the type of designs we studied. Secondly, at a fixed value of y_1, higher values for y_2 have high impact on the maximum velocity. Thirdly, by looking at the turbulence values at maximum contraction, it is beneficial to locate the point of maximum contraction far from the windward façade. Finally, by using a statistical fitting tool, we can obtain the optimum design shape, whose design can be verified by running one additional simulation.

Acknowledgements:

Part of this research was funded by GRF grant #613312, and supported by the Wind tunnel facility in HKUST.

References:

[1] Blocken, B.; van Hooff, T.; Aanen L., Bronsema, B.: Computational analysis of the performance of a venturi-shaped roof for natural ventilation: venturi-effect versus wind-blocking effect, Computers & Fluids, 48, 2011, 202-213. http://dx.doi.org/10.1016/j.compfluid.2011.04.012

[2] Building Department: "Code of practice on wind effects in hong kong," The Government of the Hong Kong Special Administrative Region, 2004.

[3] Hanjalic, K.; Launder, B.: A Reynolds stress model of turbulence and its application to thin shear flows, Journal of Fluid Mechanics, 52, 1972, 609-638. https://doi.org/10.1017/S002211207200268X

[4] Heath, M. A.; Walshe J. D.; Watson, S. J.: Estimating the potential yield of small building-mounted wind turbines, Wind Energy, 10, 2007, 271-287. http://dx.doi.org/10.1002/we.222

[5] Kolmogorov, A. N.: The local structure of turbulence in incompressible viscous fluid for very large Reynolds numbers, Doklady Akademili Nauk SSSR, 1941, 299-303.

[6] Li, Q.; Chen, F.; Li Y.; Lee, Y.: Implementing wind turbines in a tall building for power generation: A study of wind loads and wind speed amplifications, Journal of Wind Engineering and Industrial Aerodynamics, 116, 2013, 70-82. http://dx.doi.org/10.1016/j.jweia.2013.03.004

[7] Lu, L.; Ip, K. Y.: Investigation on the feasibility and enhancement methods of wind power utilization in high-rise buildings of Hong Kong, Renewable and Sustainable Energy Reviews, 13, 2009, 450-461. http://dx.doi.org/10.1016/j.rser.2007.11.013

[8] Matheron, G.: Principles of geostatistics, Economic Geology, 58, 1963, 1246-1266. http://dx.doi.org/10.2113/gsecongeo.58.8.1246

[9] Mertens, S.: The energy yield of roof mounted wind turbines, Wind Engineering, 27, 2003, 507-518.

[10] Reynolds, O.: On the dynamical theory of incompressible viscous fluids and the determination of the criterion, Philosophical Transactions of the Royal Society of London, A, *1895,* 123-164. http://dx.doi.org/10.1098/rsta.1895.0004

[11] Tominaga, Y.; Mochida, A.; Yoshie, R.; Kataoka, H.; Nozu, T.; Yoshikawa, M.; Shirasawa, T.: AIJ guidelines for practical applications of CFD to pedestrian wind environment around buildings, Journal of Wind Engineering and Industrial Aerodynamics, 96, 2008, 1749-1761. http://dx.doi.org/10.1016/j.jweia.2008.02.058

[12] www.wikipedia.org: Pearl River Tower