Upstream building effects on urban wind energy: a study of the University of Alberta North Campus

G. Cornejo , M.B Fogaing, B.A. Fleck, C. F. Lange *

Mechanical Engineering, University of Alberta, Edmonton, Canada
*carlos.lange@ualberta.ca

Abstract—This paper assesses the effects of upstream buildings on the wind energy available on the tallest building of the University of Alberta, the Donadeo Innovation Centre for Engineering (DICE). CFD simulations with $\kappa-\omega$ SST turbulence model were performed to compare the removal of upstream buildings using the same inlet boundary layer profile, which was obtained using a periodic boundary condition to ensure horizontal homogeneity through all the cases.

The results show the importance of contextualization for CFD simulations and how the upstream obstacles could significantly affect wind prediction for energy assessments. One particular building, which is 2 rows upstream but only 10 meters lower in height, has a significant effect on the approaching wind on the target location. Nevertheless, its effects are diminished by the presence of further upstream buildings. In general, the effects of the upfront buildings are perceived up to 20-30% of the height of the building from its top without an evident correlation to the number of buildings included upstream. The notable exception of the case of the tallest upstream building highlights the necessity of a framework to segregate the computational domain into building groups, reducing the computational cost of a single-building assessment and the risk of omitting relevant upstream obstacles from the simulation.

Keywords-component—Modelling urban morphology; Urban Wind; Wind energy; Complex urban geometry

I. INTRODUCTION

Electricity is one of the most essential resources in modern societies as it powers services, entertainment, and increases the productivity and quality of life in communities. For this reason, it is directly correlated with the development of nations. Unfortunately, the consequences of generating energy from fossil fuels, which have the key advantages of energy density and availability, are becoming increasingly costly. These include the artificial increase in economic cost due to carbon tax and the externalities of greenhouse gas emissions in climate change. Therefore, the interest in the use of renewable energy sources has increased substantially, encouraging the development of wind, hydro, and solar projects in general, as they can provide a sustainable alternative to traditional energy sources. Moreover, these alternatives have the added advantage of being implemented on a micro-generation scale known as distributed generation. This allows for the production of electricity at or near the point of consumption, reducing the need for long-distance transmission lines and the associated costs and energy losses. By installing small-scale renewable energy systems, such as solar panels or wind turbines, on residential and commercial buildings, communities can generate their electricity, reduce their dependence on fossil fuels, and create a more sustainable future. Studies have shown that distributed generation can contribute significantly to the renewable energy mix, especially in urban areas with high population densities, and can play a crucial role in mitigating climate change [1], [2]. Urban wind energy, in particular, can be generated from the wind passing through and around buildings, which provides an excellent opportunity to harness renewable energy in urban environments [3].

Distributed generation using renewable energy sources has become increasingly important for mitigating the negative environmental and economic consequences of traditional energy sources. Wind energy, in particular, has great potential for sustainable energy production, especially in urban environments with high population densities. However, accurately estimating the available wind energy in these environments is a complex task due to the impact of urban morphology on wind flow patterns. Computational Fluid Dynamics (CFD) is a preferred method for wind energy estimation due to its efficiency and versatility, and guidelines such as Cost Action 732 [4], Tominaga et al. [5], and Blocken [6] provide recommendations for simulating urban environments. However, despite all these guidelines and the efforts in the development of frameworks for energy assessment, the assessment of urban wind energy availability is still a challenge. One example of this is the need of accuracy in the incident wind profile [11], which is the velocity profile that is directly upstream of the obstacles.
It is essential for successful simulations, and incorporating the effects of upstream buildings on the incident flow is a delicate task, as we will explore in this paper.

Accurately estimating available wind energy in urban environments is a complex task due to the impact of urban morphology on wind flow patterns. Current urban wind energy studies often focus on determining wind flow patterns in the vicinity of the building of interest, as well as statistical characterization of winds in that location. Moreover, determining wind patterns within cities has become increasingly important for urban planning in terms of construction safety, heat island mitigation, and pollution control. While several approaches have been proposed to estimate urban wind flow patterns, including CFD simulations, accurate estimation of wind energy availability in urban environments remains a highly situational task.

CFD is the preferred method for wind energy estimation because the variability in field measurements and the scaling difficulties in wind tunnel experiments. Nevertheless, due to the complexity of the system and the amount of uncertainties, validation experiments are considered mandatory [5]. Unfortunately, experiments in urban environments are scarce because of the complex urban morphology and the time required for the experiment, which translates into significant costs in time and resources. This paper analyzes the effects of upstream buildings on urban wind energy estimations, as the presence of a variety of upstream structures is one of the main characteristics of urban environments. Upstream wind conditions are often either explicitly integrated in the simulation or modelled through roughness approximation, and it is up to the engineer to determine if the buildings are close and high enough to impact the incident flow in the building. However, it is strongly recommended to test their inclusion for accuracy [12]. Also, the accuracy of incident profiles is essential to simulation success, as it determines the turbulence and velocity magnitude of the wind that comes in contact with the building. The horizontal homogeneity of the incident profile is a known problem, as the roughness level significantly affects the expected atmospheric boundary layer profile [11]. Unfortunately, this is a common scenario in urban environments. The key variables to determine in an urban wind energy assessment are turbulence and velocity. The former has a significant effect on the life and integrity of the equipment [14], and in the system’s capability to seize the wind velocity variations, whereas the latter has a direct impact on the amount of energy directly usable for extraction [15]. This paper aims to explore some of the challenges and limitations of current approaches in estimating wind energy in urban environments, particularly the one that is associated with carefully accounting for the impact of upstream buildings on wind flow.

II. METHODOLOGY

This is a micro-scale simulation that focuses on a specific building and its surroundings. Hence, the equations are the Navier Stokes equations without Coriolis forces and with mass conservation in stationary conditions. The atmospheric boundary layer equations, used for the initialization of the developed wind profile in an empty domain, assuming neutral stratification at the inlet, corresponding to equations for the main velocity, turbulence kinetic energy and turbulence specific dissipation:

\[
U = \frac{u_{ABL}}{\kappa} \ln \frac{z + z_0}{z_0} \quad (1)
\]

\[
k = \frac{u_{ABL}}{\sqrt{\mu}} \quad (2)
\]

\[
\varepsilon = \frac{u_{ABL}^3}{\kappa}(z + z_0) \quad (3)
\]

where \(u_{ABL}\) is the friction velocity, which is determined from the equation 1 using the value of the reference velocity (5.58 m/s) at the height (80 m), both obtained from the Canadian wind atlas [6] at the building location. \(\kappa\) and \(C_p\) are the Von Karman (0.42), and turbulence kinetic energy model constants, 0.09 respectively. Finally the surface roughness, \(z_o\) value was also determined from the Canadian Wind Atlas (2m).

The geometry corresponding to the University of Alberta North Campus was obtained following a procedure from a previous work [8]. In summary, the procedure consisted in adapting a photogrammetry from 3D Warehouse, repairing discontinuities, adjusting dimensions that did not match architectural designs and defeaturing elements that are not relevant.
for the CFD simulation. The adaptation and final work of the geometry was done in SolidWorks, which allowed to have a realistic and flexible geometry to place into the domain. The domain surrounding the buildings can be seen in Figure 1. The domain was adjusted so directional blockage ratio was below the recommended 3% in all simulations.

Regarding the boundary conditions, shown in Figure 1, the specified inlets are the face of the domain upstream to the buildings and the top. The top boundary condition is set as an inlet to enable fixed velocity components, as shown in equation 1 for the incident direction and an approximation for the vertical component based on mass conservation and velocity equations. This approach avoids the need for a wall approximation and refinement at the top boundary while contributing to the horizontal homogeneity of the inlet profile. To limit the variables that affect the velocity profile to only the buildings, the bottom boundary condition (ground) is set as a no-slip wall without roughness.

To obtain the incident profile (fig 2, an empty domain was created with the same mesh as the open flow part of the original. A translational periodic boundary condition was set between the inlet and outlet. To avoid interference with the height of the first element, which is crucial for a correct boundary layer approximation in turbulence models, the inlet was pre-run on this empty domain with periodic boundary conditions until it reached a stable state. A developed profile that is reasonable for the urban wind assessment was then obtained, with a non-zero vertical velocity component kept constant and at the order of the gradient to avoid increasing residuals in continuity in the simulations. The compensation towards the fixed velocity at the top from the mass conservation equation was made because high horizontal velocity gradients appeared at the top of the domain without it, which made sense given that the gradient with respect to $z$ of the equation was not negligible on the level of accuracy.

Blocken et al. [11] highlighted the problems in CFX and Fluent when considering a height surface roughness that interferes with the height of the first element, which is key for a correct boundary layer approximation in turbulence models. To fix this, and avoid using a tremendously large domain, the inlet is pre-ran on an empty domain with periodic boundary conditions, as described above.

For the outlet condition, it is recommended to have a zero pressure gradient condition to avoid any upstream flow and subsequent problems with the convergence of the simulation. For this to work the domain has to be large enough for the downstream eddies to stabilize, $15H_{max}$, where $H_{max}$ is the height of the target building (64.3 m).

Although it was not possible to validate the present work, the conclusions will highlight the need for future validation. While it is recommended to compare different turbulence models to identify the most suitable one that represents the flow behaviour accurately [9], [10], [12], [13], this paper does not include such a comparison as it is outside its scope. Instead, the conservative $k-\omega$ SST model was chosen, which is expected to perform well compared to the standard $k-\varepsilon$ model. Since validation data is not available at present, the appropriate turbulence model can only be selected based on careful consideration of the available options.

The target building for this study is the Donadeo Innovation Centre for Engineering (DICE), which is the tallest building on campus at 64.3 meters in height, whereas the highest obstacle is the Natural Resources Engineering Facility, with a difference of roughly 14 meters. Buildings upstream were systematically removed as shown in Figure 1. The results of the simulations will be compared based on normalized velocity, with respect to the reference velocity (5.58 m/s), in comparison to the height of the building and the turbulence intensity at specific locations. Then an estimation of the average wind power density ($WPD$, Eq 4) around the entirety of the roof is compared at three different heights: 6, 10 and 20 meters above the roof to determine general effect.

$$WPD = \rho u^3$$

where $\rho$ is the air density, and $u$ the streamwise velocity. The domain was meshed according to best practice guidelines [4]–[6]. We employed 11 million elements (Fig 3). While most of the elements were hexahedrons, we utilized tetrahedrons and refinement in areas around the buildings due to their irregular shape. We ensured that all elements in the walled boundaries, such as the ground and buildings, were drawn with prismatic inflation layers. Refinements and grid convergence were determined in a previous work [8] whereas a maximum Yplus of 100 was obtained. The simulations were carried out in Ansys Fluent, utilizing the Coupled algorithm for pressure coupling, double precision and second-order discretizations. The convergence criterion was set to low residuals (1e-6) and stable velocity on monitoring points placed on the incident profile, top of the DICE Building, and at the boundary layer of the downstream part of the domain located more than 10 $H_{max}$ past DICE.

RESULTS

Normalized velocity profiles, turbulence intensity, and wind power density were estimated at the top of the DICE building, as shown in Figure 4. The effects of the surrounding buildings were found to be significant up until approximately 20-30% above the height of the buildings (approx 80m for DICE and 70 m for NREF). The results indicate that the fifth building (NREF) had a significant impact on the predictions, and, when the buildings upstream of it were removed, the increment in turbulence intensity and decrease in wind power density were significantly more pronounced, suggesting that the upstream buildings reduce the impact of the NREF on DICE’s roof.

When comparing the estimates of wind power density at the top of the DICE building with different numbers of upstream buildings, it can be observed that estimations with fewer than seven buildings are significantly lower, as shown in Table I. Another relevant observation from this result is that, in this case, the best placement of the wind turbine is not on the tallest building (DICE), but on the second tallest (NREF).
Figure. 2. Developed incident profile

Figure. 3. Mesh structure for 9 buildings

Figure. 4. Normalized velocity, turbulence and wind power density variation when different amount of buildings are considered. Cases 1 to 9 refer to 1 to 9 buildings, respectively.

Figure. 5. Normalized velocity, turbulence and wind power density on top of NREF building from 9 building simulation.
This could be explained by the fact that the latter interferes with the flow of the former in conjunction with the shape of its roof (see Figures 6 and 7), which has a favourable effect on the wind speed. These observations highlight the complex interplay between buildings and the flow of wind energy.

### Table 1

<table>
<thead>
<tr>
<th>Simulation</th>
<th>Wind Power Density [W/m²]</th>
<th>DICE Location</th>
<th>NREF Location</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td>65m 70m 80m</td>
<td>60m 70m 80m</td>
</tr>
<tr>
<td>1 Building</td>
<td>0.209</td>
<td>4.068 1.247</td>
<td>- - -</td>
</tr>
<tr>
<td>2 Buildings</td>
<td>0.778</td>
<td>5.130 3.601</td>
<td>- - -</td>
</tr>
<tr>
<td>3 Buildings</td>
<td>0.120</td>
<td>3.475 2.582</td>
<td>- - -</td>
</tr>
<tr>
<td>4 Buildings</td>
<td>0.510</td>
<td>11.974 40.412</td>
<td>- - -</td>
</tr>
<tr>
<td>5 Buildings</td>
<td>1.472</td>
<td>18.154</td>
<td>38.543 39.543</td>
</tr>
<tr>
<td>7 Buildings</td>
<td>2.087</td>
<td>18.515 38.351</td>
<td>39.825 40.790</td>
</tr>
<tr>
<td>9 Buildings</td>
<td>1.916</td>
<td>17.305</td>
<td>38.543 39.856</td>
</tr>
</tbody>
</table>

From the streamlines (in Figures 7 and 6), it can be observed that the effects of upstream buildings are not straightforward. The effect of the structures between the Natural resources (NREF) and DICE buildings might be negligible because of the presence of the former. Also, acceleration, channelling and pressure gradients might appear in the interaction of upstream building that might significantly affect the accuracy of the results.

### Conclusions and Future Work

The placement of wind turbines in urban environments is a complex task that requires a detailed understanding of the impact of buildings on wind flow. This study emphasizes the importance of considering the geometrical parameters of upstream structures and their interaction with each other, rather than solely focusing on their interaction with the target building. To achieve this, a geometrical analysis that takes into account the shape and height difference between upstream buildings is recommended.

Although this study has several limitations, it offers an opportunity for future research to improve the accuracy of wind turbine placement. For example, the use of a developed upstream profile, rather than one based on upstream roughness estimations, may result in an incident profile that has different sensitivity because it might not be fully developed. Additionally, trees and vegetation between buildings may further affect the wind flow in the same manner as the buildings upstream of NREF did in this case. The selection of the turbulence model and wind incidence variations should be carefully considered in future work when improving the current model setup. Furthermore, the absence of side buildings in this study may have hidden pressure gradients, which could increase the effect of downstream buildings and produce wind currents with significant velocities at the sides of the buildings. The main limitation of this study to increase the insight regarding the aforementioned observations is the lack of a validation study, which constrains the accuracy of the assumptions made.

Finally, this study highlights the need for a framework that can segregate the computational domain into building groups, thereby reducing the computational cost of a single-building assessment and minimizing the risk of omitting relevant upstream obstacles from the simulation. Such a framework would enable a more comprehensive and accurate analysis of wind flow in urban environments and facilitate the efficient and effective placement of wind turbines.

### Acknowledgment

G. Cornejo acknowledges the support of the Becas-Chile (ANID) scholarship - 72210240.
REFERENCES


