Impact of wind direction on wind energy potential for building-integrated ducted wind turbines: a numerical analysis

Sadra Sahebzadeh¹, Hamid Montazeri¹, Abdolrahim Rezaeiha¹,²

¹ Eindhoven University of Technology, Eindhoven, the Netherlands
² KU Leuven, Leuven, Belgium

s.sahebzadeh@tue.nl, sadrasahebzadeh@gmail.com

Abstract. The aerodynamic performance of building-integrated ducted wind turbines depends on several parameters such as the duct geometry, variation in wind speed and direction (which are inherent characteristics of the urban wind). This study focuses on the impact of wind direction on wind energy potential of a previously optimized building-integrated duct geometry [1], embedded in a generic isolated high-rise building. The mean power density at the duct throat (where the turbine can be installed) is investigated in four wind directions of $\theta = 0^\circ$, $30^\circ$, $60^\circ$ and $90^\circ$. High-fidelity steady RANS simulations, validated with experimental data, are used. The results show that the studied duct can increase the mean power density at its throat (i.e. rotor plane) up to 7.08 – 24.8 times that of the freestream flow at the same height for a wide range of $-60^\circ \leq \theta \leq 60^\circ$. The variation of wind energy potential in different wind directions is shown to be due to the increased size of the nozzle stagnation and separation regions for $\theta > 0^\circ$ which limit the nozzle effective area and lower flowrate through the throat. Flow deviation from the duct central axis towards its walls further depletes the wind energy in friction.

1. Introduction
Wind energy harvesting in urban areas can reduce the infrastructure costs as well as the losses due to long-distance energy transmission. Urban wind energy can be harvested using (i) roof-mounted wind turbines retrofitted into existing buildings, (ii) stand-alone wind turbines in urban areas or (iii) building-integrated wind turbines, where the latter can be fully integrated in the architecture of the buildings from the initial phase [2–5]. The potential of wind energy harvesting using building-integrated ducted wind turbines for high-rise buildings has been already shown in the literature [4,5] and it is known that this potential is dependent on parameters such as the duct geometry, urban setting, variation in wind speed and direction as well as local wind turbulence characteristics [6].

Hassanli et al. [7] performed a series of wind tunnel measurements to investigate the performance of different building-integrated duct geometries and found that the separation effects play a major role in their aerodynamic performance. In that study, a streamline curved duct geometry yielded the highest time-averaged streamwise velocity increase of 1.33 times the freestream velocity. Sahebzadeh et al. [1] employed Computational Fluid Dynamics (CFD) simulations to develop a neural network surrogate model describing the correlation between the building-integrated duct geometry and its wind energy potential. The optimization of the model resulted in a duct that could increase the wind power density up to 24.8 times the freestream flow at the same height. However, this study was performed at a fixed wind direction, $\theta = 0^\circ$, not addressing the inherent direction variation feature of the urban wind.
The present study aims to investigate the impact of wind direction on wind energy potential of the aforementioned optimized duct geometry. The investigation is performed using high-fidelity CFD simulations, verified and validated with experimental data. The findings of this study are believed to further clarify the potential of building-integrated ducted wind turbines, provide new insight into the impact of wind direction on their aerodynamic performance and open new avenues for optimization of these wind energy harvesting systems.

The rest of the paper is as follows: studied geometry, parameters and assessment criteria are introduced in Section 2. CFD simulations are detailed in Section 3. Solution verification and validation are presented in Section 4 followed by the results and conclusions in Sections 5 and 6 respectively.

2. Studied geometry, parameters and assessment criteria
The studied geometry consists of a generic isolated high-rise building with an optimized building-integrated duct for wind energy harvesting. The depth × width × height dimensions of the building are 30 × 30 × 150 m (W_b × D_b × H_b). The studied building-integrated duct is the result of a previous investigation in which the geometry of the duct as well as its location along the height of the building were optimized towards the highest area-weighted average wind speed at the duct throat (i.e. rotor plane). The integrated duct shape consists of a nozzle, a throat in which a horizontal axis wind turbine can be installed and a diffuser. Figure 1 and Table 1 detail the geometry of the building and the integrated duct. Further information regarding the optimization process resulting in the building integrated duct is provided in Ref. [1].

![Figure 1. (a) Geometry and dimensions of the studied building and integrated duct and (b) cross-section of the duct.](image)

| Table 1. The dimensions of the studied building-integrated duct. |
|---------------------------------------------------------------|
| **Dimension**   | **Value**   |
| Diffuser length   | 0.5W_b = 15 m |
| Nozzle length    | 0.5W_b = 15 m |
| Nozzle inlet diameter  | 0.68W_b = 20.4 m |
| Diffuser outlet diameter | 0.24W_b = 7.4 m |
| Throat diameter   | 0.1W_b = 3 m |

The impact of wind direction on wind energy potential of building-integrated ducted wind turbine is studied at four wind directions: θ = 0°, 30°, 60° and 90°. Due to the axial symmetry of the duct, the results also cover the negative values of θ = -30°, -60° and -90°. The wind energy potential is assessed based on the mean power density at the throat of the integrated duct (Figure 1), which is calculated as

\[
P_m = \frac{\bar{u}_{AW}^3 \rho}{2}
\]

where \( \bar{u}_{AW} \) is the area-weighted average of streamwise velocity at the duct throat (i.e. rotor plane) and \( \rho \) is the air density. The height of the integrated duct axis is considered as the reference height, \( H_{ref} = 135 \text{ m} \) (0.9 \( H_b \)). The reference freestream velocity, pressure coefficient and power density at \( H_{ref} \) are \( \bar{U}_{ref} = 4.35 \text{ m/s} \), \( \bar{p}_{ref} = 0.225 \text{ N/m}^2 \) and \( P_{m,ref} = 50.41 \text{ W/m}^2 \), respectively.

3. CFD simulations
3.1. Computational domain and grid
The dimensions of the full-scale 3D computational domain are selected based on the best practice guidelines for CFD simulations of wind flow in urban areas [8,9] (Figure 2a). The distance between the building and the boundaries is kept constant for all wind directions.
A computational grid with 9,117,467 cells is generated in which the bulk regions of the domain are discretized by octree hexagonal cells while polyhedral cells fill the regions close to the boundaries (Figure 2b-e). The grid resolution is based on a grid-sensitivity analysis, presented in Section 2.4. The average $y^*$ value is 230. That is within the logarithmic sub-layer of the boundary layer and is in line with requirements for the use of wall-functions, as is the case in the current study.

![Figure 2. (a) Computational domain, (b-e) computational grid.](image)

### 3.2. Boundary conditions and solver settings

Table 2 details the boundary conditions and solver settings used in the CFD simulations, which are based on the best practice guidelines [8,9].

| Boundary conditions                  |
|--------------------------------------|
| **Inlet (#1 in Figure 2a):**         |
| Atmospheric boundary layer log law   |
| profiles of mean velocity magnitude $U$ (m/s), turbulence kinetic energy $k$ (m$^2$/s$^2$) and turbulence dissipation rate $\varepsilon$ (m$^2$/s$^3$) [10]; Aerodynamic roughness length $z_0 = 0.03$ m; Friction velocity $u^* = 0.22$; Roughness constant $C_r = 7$. |
| **Outlet (#2 in Figure 2a):**        |
| Zero static gauge pressure;          |
| **Top (#3 in Figure 2a):**          |
| symmetry;                            |
| **Ground (#4 in Figure 2a), building:** |
| no-slip wall;                        |
| *For $\theta = 0^\circ$, the boundary condition of the side walls is symmetry.* |

| CFD approach                          |
|--------------------------------------|
| steady Reynolds-averaged Navier-Stokes (RANS) |

| Solver                                |
|--------------------------------------|
| ANSYS Fluent v19.1                    |

| Turbulence model                      |
|--------------------------------------|
| renormalization group (RNG) $k-\varepsilon$ with standard wall-functions |

| Pressure-velocity coupling scheme     |
|--------------------------------------|
| SIMPLE                               |

| Discretization order                  |
|--------------------------------------|
| 2nd order                             |

| Scaled residuals convergence criteria [-] |
|--------------------------------------------|
| $x, y, z$ velocity $= 10^{-7}$, continuity, $k = 10^{-9}$, $\varepsilon = 10^{-5}$ |

### 4. Solution verification and validation

A grid sensitivity analysis is performed using three consecutively refined grids with a uniform $\sqrt{2}$ refinement factor in all directions, see Table 2. The analysis is performed for $\theta = 30^\circ$ as a sample. Figure 3 shows the streamwise velocity along the central axis of the integrated duct, non-dimensionalized with $U_{ref}$. The difference between the dimensionless area-weighted average streamwise velocity values corresponding to the fine and medium grids at the throat (i.e. rotor plane) is about 0.7%, see Table 2 Therefore, the medium grid is selected for the rest of the study.

Due to the lack of high-resolution experimental data of wind flow in and around building-integrated ducted wind turbines, two sub-configuration validation studies are performed in which the CFD model is validated against wind tunnel measurements of (i) flow around an isolated building in atmospheric boundary layer incoming flow and (ii) inside a diffuser in a uniform flow. Note that the sub-
configuration validation approach has been successfully implemented in previous studies, e.g., [11]. A good agreement is achieved between the CFD results and wind-tunnel measurements in both cases. For further details regarding the validation study refer to Ref. [1].

Table 3. Grid-sensitivity analysis details and results.

| Grid    | Number of cells [-] | \(\bar{y}^+\) [-] | \(\bar{u}_{AW}/U_{ref}\) [-] |
|---------|---------------------|-----------------|-----------------------------|
| Coarse  | \(5.8 \times 10^6\) | \(\approx 310\) | 2.6                         |
| Medium  | \(9.1 \times 10^6\) | \(\approx 230\) | 2.58                        |
| Fine    | \(15.8 \times 10^6\) | \(\approx 205\) | 2.43                        |

Figure 3. Grid-sensitivity analysis: dimensionless streamwise velocity along the central axis of the building-integrated duct. Throat (i.e. rotor plane) is located at \(x/D_b = 0.5\).

5. Results

Figure 4 depicts the dimensionless area-weighted average streamwise velocity (\(\bar{u}_{AW}/U_{ref}\)) and the corresponding dimensionless mean power density (\(P_m/P_{m,ref}\)) at the duct throat (i.e. rotor plane), for different wind directions. The \(\bar{u}_{AW}/U_{ref}\) values at the rotor plane for \(\theta = 0^\circ, 30^\circ, 60^\circ\) and \(90^\circ\) are 2.9, 2.6, 1.9 and 0, respectively, corresponding to \(P_m/P_{m,ref}\) values of 24.8, 17.2, 7.08 and 0. These values show that even though the studied building-integrated duct is optimized for \(\theta = 0^\circ\), it is able to increase the mean power density up to \(\approx 7\) times the free stream for the wide range of \(-60^\circ \leq \theta \leq 60^\circ\) wind directions. As expected for \(\theta = 0^\circ\) wind direction, there is little flow entering the duct, i.e. the \(P_m\) and \(\bar{u}_{AW}\) values are close to 0.

Figure 4. Dimensionless area-weighted average streamwise velocity (\(\bar{u}_{AW}/U_{ref}\)) and corresponding dimensionless mean power density (\(P_m/P_{m,ref}\)) at the throat of the integrated duct (i.e. rotor plane). The square schematically represents the horizontal midsection of the integrated duct.

Figures 5 and 6 present the pressure coefficient and dimensionless streamwise velocity distribution along the building-integrated duct axis and at the horizontal mid-plane for different wind directions, respectively. The pressure coefficient is computed as \(C_p = (p - p_{ref}) / (0.5 \rho U_{ref}^2)\).

The pressure difference between the windward and leeward sides of the building is a driving force to direct the flow towards any opening through the building and can be a main contributing factor to the increased power density in the duct. The converging-diverging geometry of the duct can also contribute to the increased power density at the throat [1,7,6]. According to Figure 5a, pressure difference between the inlet and outlet of the integrated duct for the \(\theta = 0^\circ, 30^\circ, 60^\circ\) and \(90^\circ\) wind directions are \(\Delta C_p = 7.12, 5.66, 2.55\) and 0 respectively. These values indicate a negative linear correlation with the wind incident
angle. Figure 5b indicates a similar negative correlation between the maximum streamwise velocity at the throat (x/D_b = 0.5) and the incident wind angle. Figure 6a-c show the negative correlation of pressure difference and wind incident angle throughout the horizontal mid-plane of the duct.

Figure 6d-f shows the size of the stagnation region in the nozzle increasing with wind incident angle. Furthermore, for θ = 60° a separation bubble forms in the nozzle. The two aforementioned regions obstruct the incoming flow from entering the duct, limit the effective area of the nozzle and result in lower flowrate through the throat (i.e. rotor plane). Decreased flow rate is reflected in the lower $\bar{u}_{AW}$ values for θ > 0° in Fig. 4. The figure also shows the deviation of the mean flow to one side of the duct which will increase the friction with the walls, reducing the wind velocity at the throat. Accordingly, increased wind incident angle (θ > 0°) results in decreased mean power density at the throat.

![Figure 5](image1.jpg)

**Figure 5.** (a) Pressure coefficient and (b) dimensionless streamwise velocity along the building-integrated duct axis for different wind directions. Throat (i.e. rotor plane) is located at x/D_b = 0.5.

![Figure 6](image2.jpg)

**Figure 6.** (a-c) Pressure coefficient and (d-f) dimensionless streamwise velocity at the horizontal mid-plane of the building for different wind directions. Throat (i.e. rotor plane) is located at x/D_b = 0.5.

6. Conclusions
High-fidelity CFD simulations, validated with experimental data, are performed to assess the impact of wind direction on the wind energy potential of a previously-optimized building-integrated duct
embedded in a generic isolated high-rise building. The mean power density at the duct throat (i.e. rotor plane) is investigated in four different wind directions of $\theta = 0^\circ$, $30^\circ$, $60^\circ$ and $90^\circ$.

According to the results:

- For a wide wind direction range of $0^\circ \leq \theta \leq 60^\circ$, the studied building-integrated duct can increase the area-weighted average streamwise velocity ($u_{AW}$) at its throat (i.e. rotor plane) up to 1.9 - 2.9 times the freestream velocity. The minimum and maximum correspond to $\theta = 60^\circ$ and $0^\circ$, respectively. This corresponds to 7.08 – 24.8 times increase in mean power density compared to the freestream flow (Figure 4).

- The variation of wind energy potential in different wind directions is due to the increased size of the stagnation region in the nozzle for $\theta > 0^\circ$ and the appearance of separation bubbles for $\theta \geq 60^\circ$ which limit the effective area of the nozzle and result in lower flowrate through the throat. Furthermore, for $\theta > 0^\circ$ the flow deviates from the central longitudinal axis of the duct towards the walls which depletes the wind energy in friction (Figure 6).

The findings point out to the high potential of building-integrated ducted wind turbines in increasing the available wind power density in a wide range of wind directions. This ability can be of especial interest in urban areas with frequent variation in wind direction and velocity amplitude. The provided insight into the impact of wind direction on the wind energy potential of building-integrated ducted wind turbines is also essential for understanding their aerodynamics.

Acknowledgements

The 3rd author is a postdoctoral fellow of the Research Foundation – Flanders (FWO) and is grateful for its financial support (project FWO 12ZP520N). The authors acknowledge the partnership with ANSYS. This work was carried out on the Dutch national e-infrastructure with the support of SURF Cooperative.

References

[1] Sahebzadeh S, Montazeri H, Rezaeiha A. Building-integrated ducted wind turbines for wind energy harvesting: CFD simulation, neural network modelling and optimization. Under review.
[2] Sahebzadeh S, Rezaeiha A, Montazeri H. Towards optimal layout design of vertical-axis wind-turbine farms: Double rotor arrangements. Energy Convers Manag 2020;226:113527.
[3] Rezaeiha A, Montazeri H, Blocken B. A framework for preliminary large-scale urban wind energy potential assessment: Roof-mounted wind turbines. Energy Convers Manag 2020;214:112770.
[4] Micallef D, Van Bussel G. A Review of Urban Wind Energy Research: Aerodynamics and Other Challenges. Energies 2018;11.
[5] Anup KC, Whale J, Urmee T. Urban wind conditions and small wind turbines in the built environment: A review. Renew Energy 2019;131:268–83.
[6] Li W, Li S. Investigation of Factors Influencing Power Takeoff for a Wind Turbine Installed Inside a High-Rise Building. J Energy Eng 2020;146:4020021.
[7] Hassani S, Chauhan K, Zhao M, Kwok KCS. Application of through-building openings for wind energy harvesting in built environment. J Wind Eng Ind Aerodyn 2019;184:445–55.
[8] Franke J, Hellsten A, Schlunzen KH, Carissimo B. The COST 732 Best Practice Guideline for CFD simulation of flows in the urban environment: a summary. Int J Environ Pollut 2011;44:419–27.
[9] Tamura T, Nozawa K, Kondo K. AIJ guide for numerical prediction of wind loads on buildings. J Wind Eng Ind Aerodyn 2008;96:1974–84.
[10] Richards PJ, Hoxey RP. Appropriate boundary conditions for computational wind engineering models using the k-epsilon turbulence model. J Wind Eng Ind Aerodyn 1993;46:145–53.
[11] Alsailani M, Montazeri H, Rezaeiha A. Towards optimal aerodynamic design of wind catchers: Impact of geometrical characteristics. Renew Energy 2021;168:1344–63.