

Evaluation of an Integrated Roof Wind Energy System for urban environments

B Patankar, R Tyagi, D Kiss* and A B Suma

IBIS Power, Klokgebouw 112, 5617 AB Eindhoven, NL

*Corresponding author: diana@ibispower.eu

Abstract. Integrating renewable energy in the urban environment is of importance for the renewable energy goals set by the European Union. This research is to study and evaluate wind energy potential for an Integrated Roof Wind Energy System on the rooftop of the buildings of different heights and in different locations with the help of numerical modelling (CFD). The Navier-Stokes equations are solved using the SIMPLE algorithm while the turbulence is modelled using the $k-\omega$ SST equations. All the simulations are performed using OpenFOAM. Results shows that wind speed can be accelerated by ~ 1.4 times till it reaches the periphery of the turbine inside the unit, which will increase wind power output considerably. This results in power factor increase of 1.7 for tall buildings. Therefore, enabling combined micro wind and solar energy systems to be a viable option for urban environments.

1. Introduction

The EU's renewable energy directive [1] has set a target of achieving 20% of final energy consumption from renewable energy sources by 2020, wind and solar energy are bound to play a pivotal role in it. Among the problems with solar and wind energy, the intermittent nature of the energy source is an obstacle in steady energy harvesting. Combining of different energy sources is a probable solution to overcome this shortcoming. When it comes to the urban environment, low wind speed and lack of surface area available for solar panels are main obstacles. Integrated Roof Wind Energy System [2] or PowerNEST [15], aims to provide a combined solution of wind and solar energy, tackling the problem of low wind speeds along with large space available for placing Solar PVs. This work evaluates the wind energy potential of PowerNEST for different urban conditions (building height and surroundings).

2. Integrated Roof Wind Energy System

PowerNEST is a unique implementation of the venturi effect [3] on top of buildings in urban environments [4], idea behind the design is to capture the wind flowing over top of buildings and lead funnel influenced accelerated wind to a Vertical Axis Wind Turbine (VAWT). Figure 1 illustrates the concept of the PowerNEST.



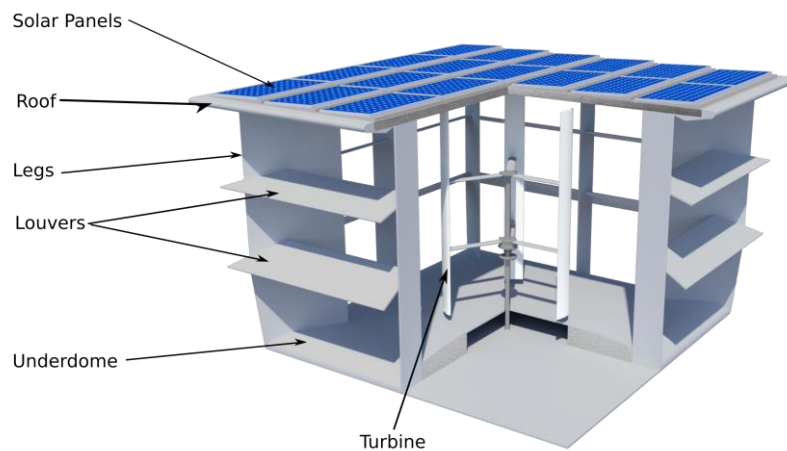


Figure 1: PowerNEST concept

Unit design is based on the guidelines described in the internal report [15], where main elements, Figure 2, are the lower & upper louvers (black), vertical louvers (red), roof (purple), legs (green) and the underdome (blue). Wind flow near top edge of high buildings has large vertical flow component, louvers capture it, while roof, underdome and legs together accelerate the wind and direct it towards the turbine.

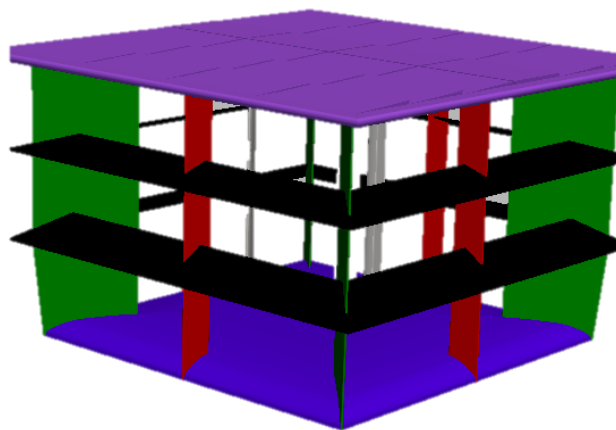


Figure 2: PowerNEST: Various unit elements are indicated in different colours

Henceforth, PowerNEST, complete with all the elements and the turbine will be referred to as “the unit” and the building on which it is to be placed is referred to as “the building”. A 3-bladed VAWT of 3.6 kW rated power capacity having 3.6m & 1.2m height and radius respectively, where blades have NACA 2412 aerodynamic profile, is considered rigid throughout this study.

3. Case Studies

To evaluate the wind energy potential of the unit in different locations, sites were chosen based on whether they were coastal open (near the coast, relatively less construction/vegetation), coastal urban (near the coast, populated urban areas), inland open (inland, less vegetation and habitation) or inland urban (cities, inland). The locations chosen based on these categories are:

- Coastal Urban – Busitel,
- Coastal Open – Bella Center,
- Inland Urban – Urkhovenseweg, Andromedaplaats,
- Inland Open – Brabant Water.

Details about dimensions of these buildings are given in the Table 1.

Table 1: Building details

No	Building	Location	Width(m)	Length(m)	Height(m)
1	Busitel	Amsterdam	21.5	92.4	43.3
2	Urkhovenseweg	Eindhoven	85.2	20.8	39.2
3	Andromedaplaats	Eindhoven	56	57.5	41.7
4	Bella Center	Copenhagen	150	120	10
5	Brabant Water	Zevenbergen	58.4	37.6	10

Selected locations deemed to be potential sites for the placement of unit(s). As one can see from the Table 1, the locations cover a wide range of heights, thus making a broader evaluation.

4. Methodology

4.1. Geometry

The building, unit and the surroundings are considered for the simulations. Detailed 3D models of the building, unit and surroundings are prepared with a 1:1 scale in SOLIDWORKS [5]. The building geometry is simplified by omitting the smaller details like façade design etc. for the ease of mesh generation. Inclusion of the surroundings of the building needs to be done in case, the surroundings appear to impact the flow on the building. The turbine details are obtained from the manufacturer. The case presented here is the flow over Busitel. The procedure is same for rest of the buildings considered in this study. The building details for Busitel are as shown in

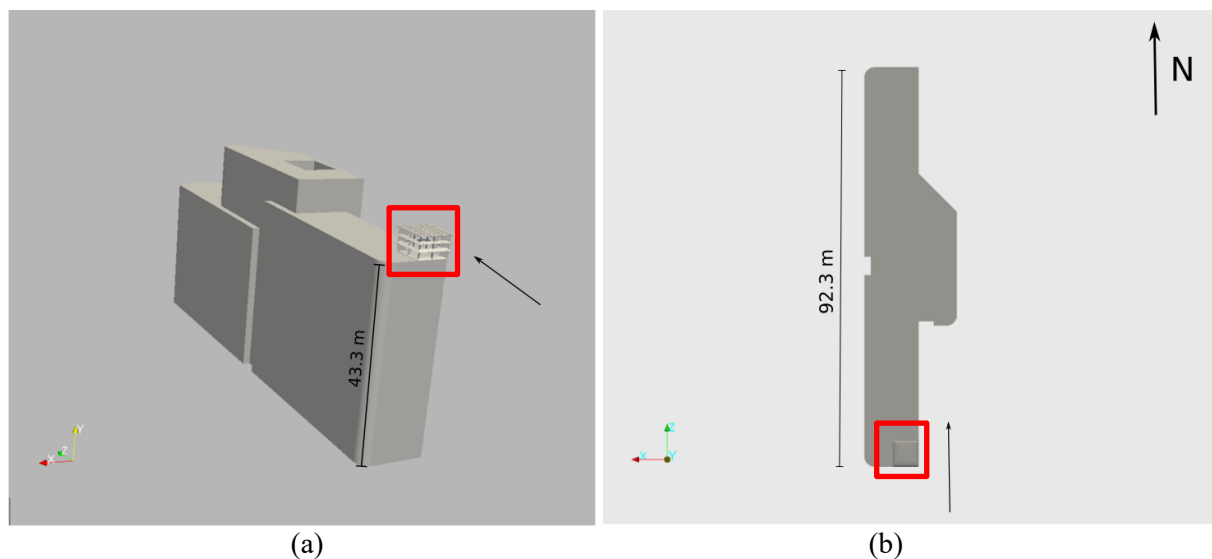


Figure 3: Busitel building with unit: Incident wind direction (South) is shown with arrow;
(a) Isometric View (b) Top View

4.2. CFD Setup

4.2.1. Meshing

Unstructured meshes are generated using the inbuilt tool – snappyHexMesh [11]. The domain bounds are important parameters that determine the accuracy of the solution. Size of the domain in flow direction must be large enough to ensure fully developed flow and wake effects can be captured. Similarly, domain perpendicular to flow-direction, must be large enough for freestream boundary conditions to be applicable. As per the recommendations [17], a blockage ratio of less than 3% was

ensured. Also, width, height and length of the domain were always 7, 5 & 15 times more than the respective building dimensions.

Capturing smaller but important geometrical features and their influence on flow dynamics while ensuring less cell counts, require to have local surface and volume refinements. And refinement levels for both surfaces and zones are used as per geometrical and flow behaviour in particular region. To include the rotational effects in the mesh, an additional zone (Figure 4) is created around the turbine for which rotation speed will be specified.

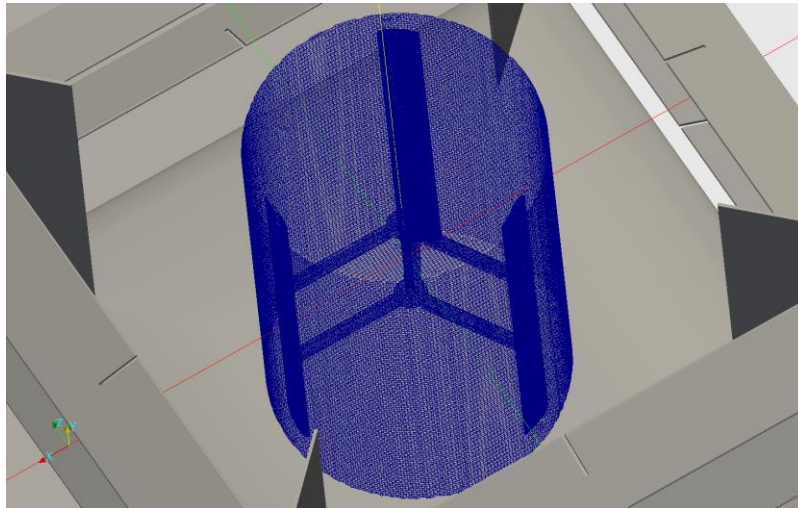


Figure 4: Mesh zone around the turbine

Three different meshes were generated to study the mesh independency of the solution. Resolution of the meshes was changed by varying the degree of local refinement levels. However, the surface refinement on the various entities was kept same as it is necessary to capture important geometric features. On an average the cell size near important entities like blades, is ~7mm, details about various meshes are shown in the Table 2.

Table 2: Details of different meshes

Type of mesh	Local Refinement Level	Cell Size (m)	Total number of cells
Coarse	5	0.125	~ 7 million
Medium	6	0.0625	~ 11 million
Fine	7	0.03125	~ 25 million

Steady state simulations were performed for a case with above mentioned meshes. Comparison between averaged wind speed on the planes as indicated in Figure 7 are presented in following figures.

Figure 5 shows wind speed vs mesh resolution on Plane – 1, before the enclosure, i.e. unit inlet with and without rotational effects, whereas Figure 6 is for Plane – 2 which is after legs, i.e. just before turbine. There is little variation (~2 %) in the wind speed values as we move from a coarse to fine mesh, for both with and without rotating simulations. Given the computational time constraints, it was decided proceed with the medium mesh for further study.

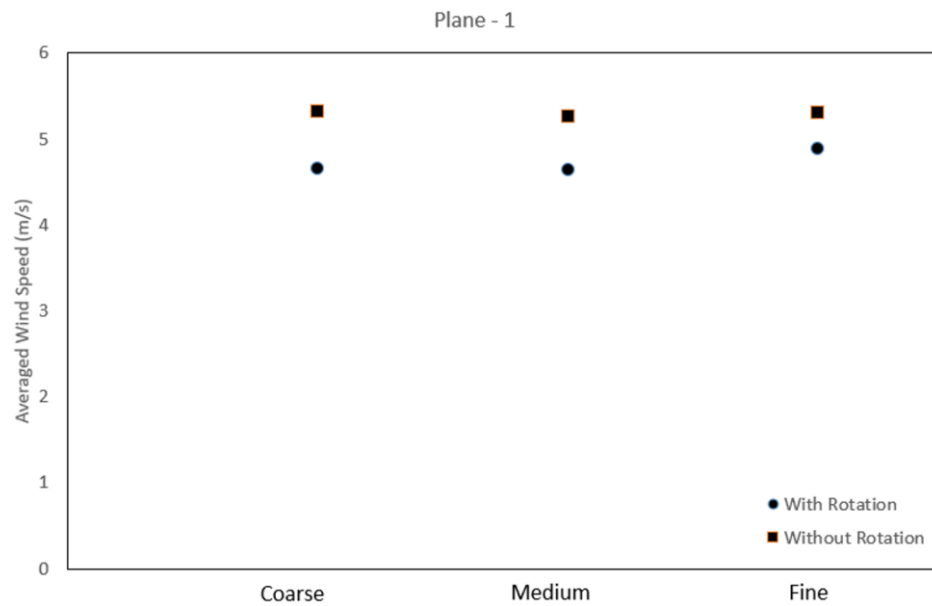


Figure 5 : Comparison of averaged wind speed on Plane - 1 with and without rotational effects

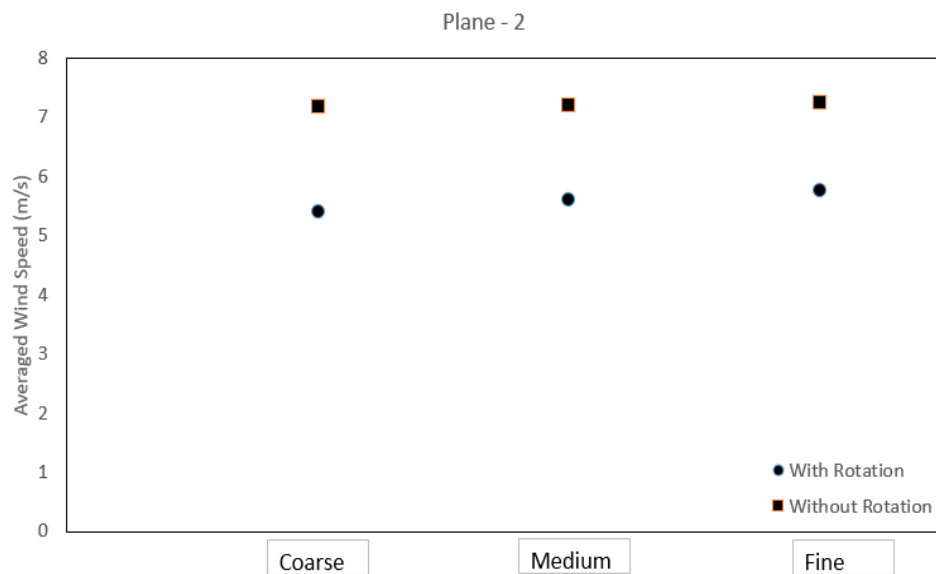


Figure 6 : Comparison of averaged wind speed on Plane - 2 with and without rotational effects

4.2.2. Boundary Conditions

The building, unit and turbine surfaces along with ground surface, are treated as no slip walls. The atmospheric boundary condition described in 4.2.2.1 section is applied on the domain inlet while outlet is treated as a pressure outlet surface. Remaining surfaces, being sufficiently far, are defined as free – stream.

4.2.2.1 Atmospheric boundary layer condition (ABL Condition)

The effect of the surroundings can be included by specifying the appropriate surface roughness parameters for different locations. The atmospheric boundary condition as implemented by Hargreaves and Wright [6] is used as it incorporates the effect of surface roughness in the velocity assignment, a logarithmic velocity profile, equation 1, is specified at the inlet. This velocity profile reads as follows:

$$U = \frac{U^*}{\kappa} \ln \left(\frac{z - z_0 + z_g}{z_0} \right) \quad (1)$$

where z_g is the minimum value of z coordinate and U^* is the friction velocity given by

$$U^* = \kappa \frac{U_{ref}}{\ln \left(\frac{z + z_0}{z_0} \right)} \quad (2)$$

where, κ is von Karman constant, z_0 is surface roughness length, and value of z_0 [14] is based on building location if it is coastal open/urban or inland open/urban.

The turbulent kinetic energy, k , profile at the inlet is as follows:

$$k = \frac{(U^*)^2}{\sqrt{C_\mu}} \quad (3)$$

where, C_μ is the model constant (=0.09).

In the case of turbulent dissipation, ω , inlet is prescribed assuming a constant value of velocity. With enough iterations one can suffice that omega converges to its steady state value.

Usage of this boundary condition assumes, no vertical velocity, constant shear stress in the boundary layer, constant pressure in vertical and stream-wise direction and conservation of the turbulence quantities.

For the present study, inlet being sufficiently far away from any elements that might influence the flow vertical component of the velocity will be zero. Since density variation with height is neglected, pressure variation along the height can be neglected. All the conditions being satisfied this boundary condition can be specified for all the cases considered.

A summary of the boundary conditions applied for the different variables are summarized in Table 3.

Table 3: Boundary conditions for different surfaces

	Pressure (p)	Velocity (U)	Turbulent Kinetic Energy (k)	Turbulent Dissipation (ω)
Inlet	zeroGradient	Atmospheric Boundary Condition	Atmospheric Boundary Condition	Constant value
Outlet	zeroGradient	zeroGradient	zeroGradient	zeroGradient
Walls	zeroGradient	No Slip , $U = 0$	zeroGradient	zeroGradient
Freestream	zeroGradient	zeroGradient	zeroGradient	zeroGradient

4.2.3. Flow and Turbulence Modelling

The *SST k- ω* model developed by Menter [8] essentially blends the *k- ω* model in the region near the wall and *k- ϵ* model in the freestream resulting in better handling of flows with adverse pressure gradients. This model has been successfully used for turbulence modelling of VAWT's [17]. Lam and Peng [17], Bachant and Wosnik [20] have studied the wake characteristics of vertical axis wind turbines. Both studies have compared the *SST k- ω* model with DES and Spalart-Allmaras [19] models respectively. Lam and Peng obtain a good agreement of the wake characteristics for 3D and 2D simulations using *SST k- ω* model when compared to stereoscopic PIV data of Tescione, et.al. [18]. However, they do observe an over-prediction of axial flow velocity in 2D simulations attributed to the infinite blade depth and tower height. Bachant and Wosnik conclude that *SST k- ω* model was effective

in predicting mean velocity fields as well as the turbulent kinetic energy, however they do mention about over-prediction of power coefficient with *SST k- ω* model.

With the high level of refinement on the surface of the enclosure as well as the turbine blades, y^+ values <30 were calculated on certain elements. Considering the above studies, the *SST k- ω* closure for turbulence modelling with the RANS approach is deemed to be most appropriate for the current cases. The simulations are done using the open source software - OpenFOAM. Rotational effects were included using the MRF approach [13]. Steady state simulations were performed assuming that the wind turbine rotate at constant speed for a given inlet velocity, it was determined by assuming a tip-speed ratio of 2.

4.3. Post processing

The post processing of the CFD results are done using ParaView [12]. To get an idea about flow patterns within the unit, contour plots of velocity magnitude are generated on a horizontal plane between (middle) upper and lower louvers and slicing it along its length.

Relative acceleration is measured to gain an idea about possible wind acceleration because of unit. This is measured by computing area weighted average velocity at different planes as shown in the Figure 7, plane-1 is 0.4 m upstream from the underdome edge and the plane-2 is just after the legs.

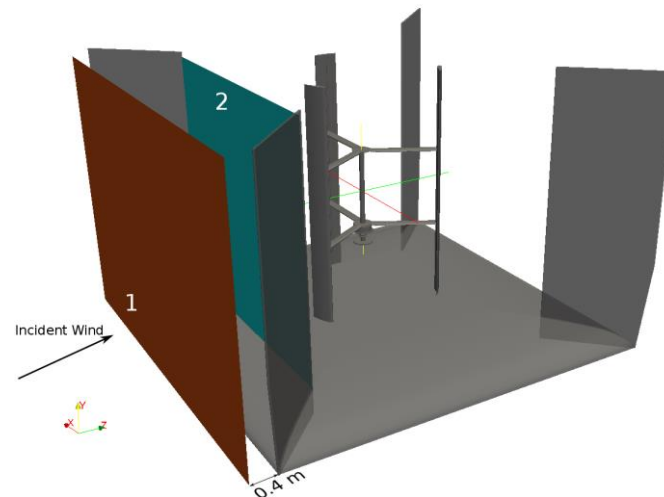


Figure 7: Post-processing planes; arrow shows incident wind direction: 1-before unit, 2-after legs

The area weighted average velocity is calculated in 2 steps, initially by computing the velocity magnitude on the plane as,

$$U_{mag} = |U| = \sqrt{(U_x^2 + U_y^2 + U_z^2)} \quad (4)$$

and then area weighted average is computed as follows:

$$U_{avg} = \frac{\sum_{i=1}^M |U|_i A_i}{\sum_{i=1}^M A_i} \quad (5)$$

where A_i is the area of cell with index i and $|U|_i$ is velocity magnitude in the cell i . Relative acceleration is calculated as the ratio of U_{avg} at planes 2 & 1 as indicated in Figure 7.

$$\phi = \frac{U_{avg2}}{U_{avg1}} \quad (6)$$

Steady state MRF simulations can be used to estimate time averaged values of pressure and velocity fields around the wind turbine. Thus enabling calculation of steady state values of force and torque on

the blades of the turbine and thereby the steady state values of power output. To evaluate the expected power output from the turbine using CFD simulations, total moment ($\vec{\tau}$) on turbine blades along a specified vector $\vec{r}$ was calculated in the following manner:

$$\vec{\tau} = \vec{r} \times \vec{F} \quad (7)$$

where $\vec{F}$ is the specified force vector, which is sum of the pressure and viscous force components, and then power output is calculated as,

$$P = \tau_y \omega_R \quad (8)$$

where τ_y is the y – component of the torque and ω_R is the rotational speed in *rad/s*.

5. Results

5.1. Wind Measurement

Wind measurements indicate the dominant wind direction and average wind speed over a certain interval. Knowledge of the dominant wind direction and speed is crucial to gain an idea of the incident wind on the unit. In certain cases, wind measurements are done locally by placement of a 3-D anemometer at the desired location. Wind speed and directions are measured for a period of 2-3 months at a frequency of 1Hz. This data is then averaged to obtain hourly and daily basis wind measurement. In cases where measurements were not possible, data from openly available sources [9,10] are used, collected data is usually put in the form of wind rose for particular location.

Figure 8 shows the wind rose for the Busitel building in Amsterdam. Data obtained from [10], indicates dominant wind for WSW & SSW direction with a yearly average of 6 m/s. Unless specified these measurements are performed at a height of 10m which used as an input for CFD simulations. The inlet boundary condition adjusts for the variation of wind speeds with the height as per ABL Condition, and current study is for south wind.

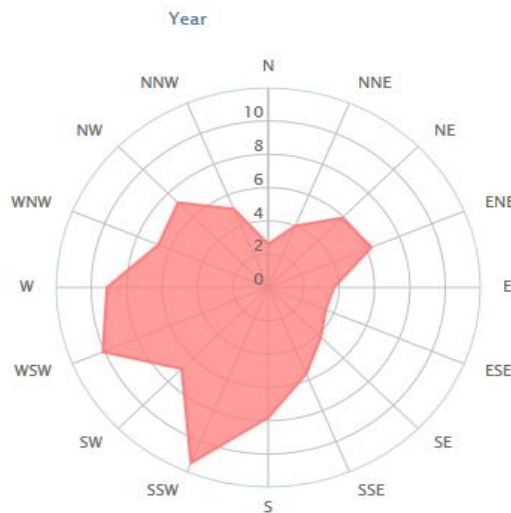


Figure 8 : Wind Direction Distribution (%) for the location Busitel [10]

5.2. Unit Evaluation

Following sections discuss the evaluation of the unit on Busitel building with wind incident from the south, and then comparison with other buildings. Simulations were performed till steady state solution is converged with residual values between 10^{-3} and 10^{-4} .

5.2.1. South Direction – Acceleration of the wind

Figure 9 shows the velocity contours plots for a horizontal plane between (middle) upper and lower louvers, with and without rotational effects.

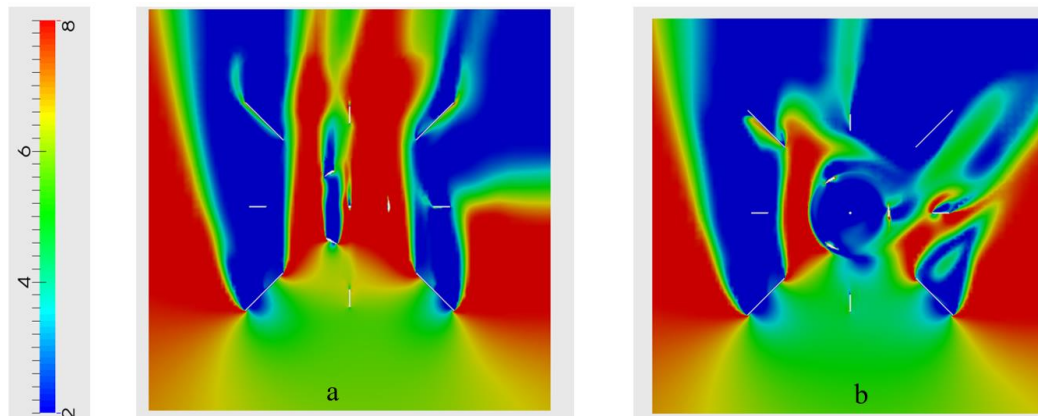


Figure 9: Velocity plots on a horizontal cross-section between louvers @ 6m/s far field wind speed:
(a) Without rotational effects (b) With rotational effects

Wind speed acceleration can be clearly seen from the plot when rotational effects are not included. From the contours one can see that the inlet wind speeds to the unit are roughly in the order of 5-6 m/s while the wind speed near the turbine location is ~7-8 m/s. It becomes little complicated when rotational effects are included. Rotating turbine influences the flow patterns at the inlet as well as turbine's surrounding. With higher rotational speed, flow tends to divert (Figure 10) from the unit because of higher resistance to the flow entering enclosure, causing the wind to escape from the edges.

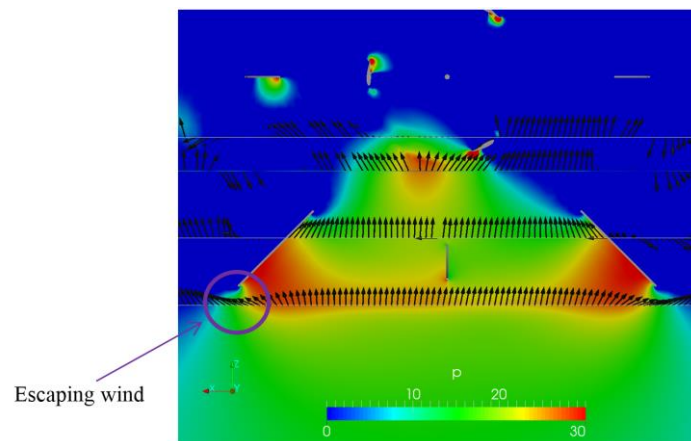


Figure 10: Pressure contour with velocity vectors on a horizontal plane in mid of the unit @ 6m/s far field wind speed

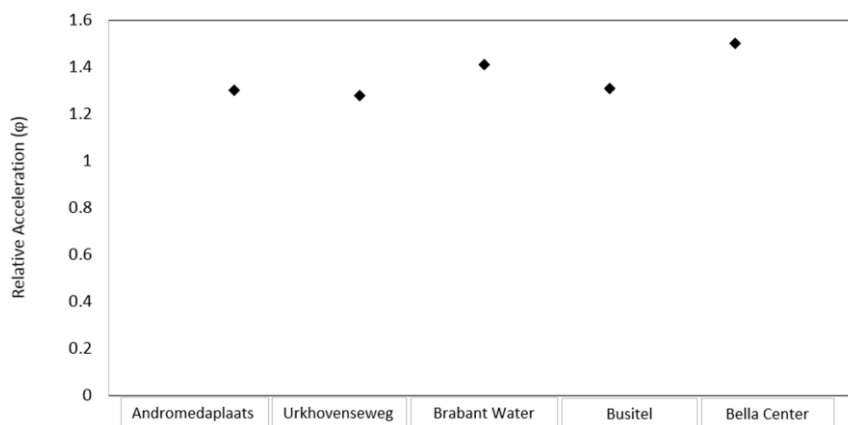


Figure 11: Relative wind acceleration for different buildings

In both cases, relative acceleration is evaluated for all the locations. Figure 11 indicates achieved relative wind acceleration, an average acceleration of ~ 1.4 times is achieved when rotation effects are not considered.

5.2.2. Power Output

Figure 12 shows the steady state power output from CFD calculations. These values are obtained from the post processing done as in Eq. (8). Comparison is done with power estimation of a standalone turbine at 6 m/s with an overall efficiency of 30%. A factor of increase in power is estimated as

$$\phi = \frac{\text{Calculated Power from CFD} - \text{Estimated Power for standalone turbine}}{\text{Estimated Power for standalone turbine}} \quad (9)$$

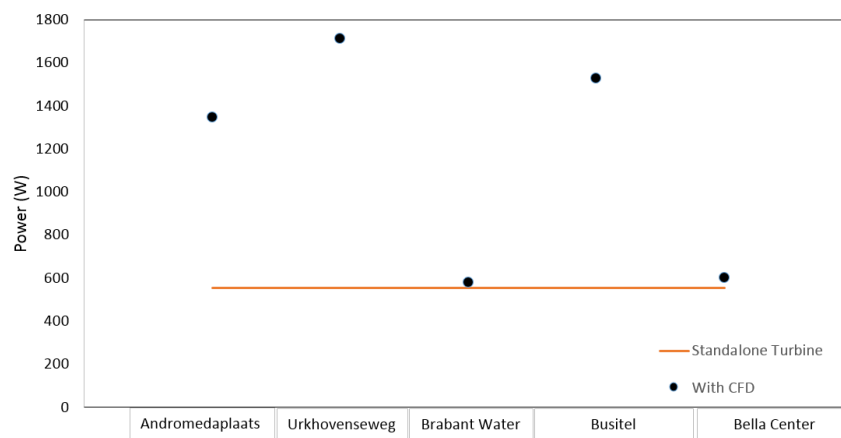


Figure 12: Power output from CFD calculations

As one can see, for the power output and correspondingly the power factor is higher for the taller buildings. As noted in Table 4, on an average a power factor increase of ~ 1.7 is estimated for taller buildings. This factor roughly corresponds to an increase of ~ 1.4 times in the wind speed reaching the turbine. Higher power factor values for taller buildings, shows better unit performance with building height. Validation of these results however is still ongoing and require experiments performed on the unit. A summary of the power output is presented in Table 4.

Table 4: Power Output summary and comparison with standalone turbine

Location	Relative Acceleration	Estimated Power (W)	Factor of increase in Power (ϕ)
Andromedaplaats	1.3	1346	1.43
Urkhovenseweg	1.28	1712	2.09
Brabant Water	1.41	579	0.05
Busitel	1.31	1529	1.76
Bella Center	1.5	603	0.09
Standalone Turbine	1	554	Base

6. Conclusions

The article aims to evaluate the wind potential of PowerNEST for different locations. In this study, results for buildings on different locations in Europe are presented, wind measurements performed to measure the variation of wind speeds and directions at the location of the unit. CFD studies done to evaluate the performance of the unit as a whole, which shows an average acceleration of factor 1.4 with

respect to the inlet. A power factor increase of 1.7, is achieved for the cases when buildings are tall. The main purpose of the PowerNEST is to capture the low winds and accelerate it to get better power outputs seems to be fulfilled in these studies. This makes micro wind energy generation as a potential solution for the urban power generation. However, these results are preliminary CFD results and need to be rigorously validated with experiments. The MRF method is an approximation, and unsteady phenomena cannot be captured with this approach. Detailed simulations studying the unsteady flow patterns within the unit will shed light on the interaction of different elements of the unit and the turbine. This is listed to be done as a future work. With the unsteady simulations, time and angle variation of torque on the blades can be studied in much detail, thereby enabling study of effects like dynamic stall of the wind turbine. Experimentation and validation of the concept at the installation sites are the further steps in completing the evaluation of the design and will provide insight into optimisation of the design.

References

- [1] Suma, A.B. "Wind Energy System," U.S. pat 20100247302, University of Miami, 2010.
- [2] <https://ec.europa.eu/energy/en/topics/renewable-energy> as accessed on 24-02-2016
- [3] White, F. M. "Fluid mechanics" *McGraw-Hill, New York, NY, USA*: 3-225, 1999.
- [4] Suma, A.B., et al. "Integrated roof wind energy system." EPJ Web of Conferences. Vol. 33. EDP Sciences, 2012.
- [5] SolidWorks, Introducing. "Solidworks corporation." *Concord, MA*, 2002.
- [6] Hargreaves, D. M., and Wright, N. G. "On the use of the k- ϵ model in commercial CFD software to model the neutral atmospheric boundary layer." *Journal of Wind Engineering and Industrial Aerodynamics* 95.5 (2007): 355-369., 2007
- [7] Patankar, S. "Numerical heat transfer and fluid flow". CRC press, 1980.
- [8] Menter, F R. "Improved two-equation k- ω turbulence models for aerodynamic flows.", 1992.
- [9] Koninklijk Nederlands Meteorologisch Instituut. <http://www.knmi.nl/home> As accessed on 09-06-2016.
- [10] Windfinder windrose data. <https://www.windfinder.com/windstatistics/sluselobet> As accessed on 09-06-2016.
- [11] Open CFD "OpenFOAM programmer's guide." *OpenFOAM Foundation* 2.0 (2011).
- [12] Ayachit, U., The ParaView Guide: A Parallel Visualization Application, Kitware, 2015, ISBN 978-1930934306
- [13] Luo, J. Y., and Gosman, A.D.. "Prediction of impeller-induced flow in mixing vessels using multiple frames of reference." INSTITUTE OF CHEMICAL ENGINEERS SYMPOSIUM SERIES, 1994.
- [14] The Swiss Wind Power Website. www.wind-data.ch/tools/profile.php?lng=en As accessed on 10-06-2016
- [15] Tyagi, R., Kiss, D. and Suma, A.B. PowerNEST 6x6 unit optimization implementation, 2016. (Internal Report)
- [16] Tominaga, Y., et al. "AIJ guidelines for practical applications of CFD to pedestrian wind environment around buildings." *Journal of wind engineering and industrial aerodynamics* 96.10 (2008): 1749-1761.
- [17] Lam, H. F., and H. Y. Peng. "Study of wake characteristics of a vertical axis wind turbine by two- and three-dimensional computational fluid dynamics simulations." *Renewable Energy* 90 (2016): 386-398.
- [18] Tescione, G., et al. "Near wake flow analysis of a vertical axis wind turbine by stereoscopic particle image velocimetry." *Renewable Energy* 70 (2014): 47-61.
- [19] Spalart, Philippe R., and Stephen R. Allmaras. "A one equation turbulence model for aerodynamic flows." *AIAA journal* 94 (1992).
- [20] Bachant, Peter, and Martin Wosnik. "Modeling the near-wake of a vertical-axis cross-flow turbine with 2-D and 3-D RANS." *arXiv preprint arXiv:1604.02611* (2016).