Optimal Design of Building for Urban Wind Energy Utilization

Olga Hubova 1, Marek Macak 1
1 Slovak University of Technology, Faculty of Civil Engineering, Radlinskeho 11, 811 05 Bratislava, Slovak Republic

olga.hubova@stuba.sk

Abstract. The article deals with numerical and experimental investigation of wind flow in roof area and optimal design of input parts, where is possible to use IRWES system (Integrated Roof Wind Energy System). Orientation of some high-rise objects is suitable for using wind power. In the first phase, the selected building was investigated using CFD simulation for creating space for three small wind turbines in the area of two technical floors. By using 3D print technology, model of the structure in scale 1:300 with rough façade was created. Experimental measurements were performed in Boundary Layer Wind Tunnel in Bratislava. Measurements were made for 3 reference wind speeds, which fulfilled flow similarity of prototype and model. We have compared the results of the numerical simulations and experimental measurements and obtained information on the average wind speeds at the VAWT site. Comparison of the mean wind velocity and external wind pressure coefficient obtained by CFD simulation and experimental measurements showed a good match. Considering the annual average wind speed at about 100m above sea level, we compared the wind acceleration at the turbine site.

1. Introduction
Wind energy conversion has experienced a growth worldwide in the last twenty years. Building augmented wind turbines – roof installations in urban environment are in new research activities. VAWT - Vertical Axis Wind Turbine technology offers advantages, as low noise see [1] and [2].

In Bratislava, broader central zone began intensive construction of high-rise building with total height of 90 to 125m as is possible to see in figure 1. For buildings orientated towards the predominant wind, optimal roof design according to prevailing wind direction in locality, should give us distribution of mean wind velocity suitable for using wind energy.

2. Methodology
In the first phase, the selected building was investigated using CFD simulation in the ANSYS program, where we will deal with the mean wind velocity distribution inside the upper part of the selected building see figure 2, where small wind turbine should be placed. Average wind speeds measured over the last 10 years by the Slovak Hydro Meteorological Institute give good information about the city's wind conditions (see figure 3).

At the top of the selected object, we tried to create space for 3 small wind turbines where geometry of the model was derived from the existing building with specific façade, see figure 2. Cross section of
the building is elliptic with dimension 48 × 27 m and high of building is 105m. Possible free space in top of the building is on two technical floors.

![Figure 1. Northwest view of future development in Bratislava city](image1.png)

![Figure 2. Model of selected high-rise building](image2.png)

![Figure 3. The frequency of wind directions in parts per thousand and mean wind velocity [m/s] in Bratislava city](image3.png)

We tried to find, using numerical simulation, optimal design of the wind turbine inlet/outlet space on the top of high-rise building. Next step were experimental measurements on scale model of building performed in Boundary Layer Wind Tunnel (BLWT) of Slovak University of Technology in Bratislava.

3. Numerical simulation

The main goal of CFD simulation was to determine mean wind velocity distribution in the modified free space in the upper part of the building which allows acceleration of wind speed and provides sufficient speed to obtain wind power. For the analysis of our problem we chose the finite volume method implemented into program ANSYS Fluent [10], which offers several turbulence models.

3.1. Description of the top of building
Possible free space in top of the building is on two technical floors in roof of the building. There are possibilities to integrate three wind turbines. Cross-section of the middle section is $5 \times 12$ m and $5 \times 10$ m for side section see figure 4.

![Figure 4. Empty space of facade of the structure](image)

### 3.2. Computation domain and generated mesh

Model of building was created according to an existing building. The computational domain has size $1\times0.6\times0.4$ km$^3$ ($l\times w\times h$), corresponding with block ratio 2%, which is below the recommended maximum value of block ratio 3% [8, 9]. The distance from the object to the inlet was 400m.

We created mesh using Meshing implemented in ANSYS [10]. The size function was set on proximity and curvature with medium relevance centre. The element size on surface of the building was 1 m with the size function set on curvature. The element size on surfaces of the designed wind turbine inlet/outlet was 0.25 m with the size function set on curvature. 2 071 521 elements with 387 717 nodes which were transformed in Fluent [10] to polyhedral mesh with 455 303 cells, 2 895 448 faces and 2 376 340 nodes were generated.

### 3.3. Numerical model, boundary conditions and solver setting

For the solution of the 3D steady RANS equations with standard $k$-$\varepsilon$ model [11] we used CFD code ANSYS Fluent [10]. For near-wall treatment, the standard wall functions by Launder and Spalding [11] were used. The inlet boundary conditions of the domain are defined by the vertical profiles

$$v(z) = \frac{v^*}{\kappa} \ln \frac{z + z_0}{z_0}, \quad (1)$$

where $v(z)$ is mean wind velocity at height $z$, $v^*$ is shear velocity, $z_0$ is aerodynamic roughness height ($z_0=0.7$), $\kappa$ is von Karman constant ($\kappa = 0.42$), $v_{ref} = 10.98467$ m/s at a reference height $z_{ref} = 100$ m.

Additional inputs for $k$-$\varepsilon$ model are equation for turbulent kinetic energy $k$, and turbulence dissipation rate $\varepsilon$ as follows:

$$k = \frac{u'^2}{\sqrt{C_\mu}}, \quad \varepsilon(z) = \frac{u'^3}{\kappa(z + z_0)}, \quad (2)$$

where $C_\mu = 0.09$ is a model constant.

The outlet boundary is defined as pressure outflow and the side and upper boundary as zero gradients (symmetry). The bottom of the computational domain is modelled as a slip wall. All computations were run as pressure-based, steady, without production limiter. As the solution method SIMPLE pressure-velocity coupling scheme with second order spatial discretization was used, for transient formulation was used second order implicit method. Solution was initialized by hybrid initialization with default setting.

In the following figure 5, one can see the pressure and wind velocity distribution on the building and streamlines near the top of the object.
3.4. Design of inlet/outlet shape on the top of building

According to numerical simulation we designed symmetric shape of the inlet/outlet wind turbine space 3 × 3m. For modelling of the surfaces, we used spline. Side surfaces were created with tangent 45° on outer end and 0° tangent in centre. The bottom surface was modelled using spline with 0° tangent on both ends. The upper surface was flat, see Fig. 6.

Wind pattern for designed wind turbine inlet/outlet space for wind direction 90° can be seen in Figure 7.
4. Experimental measurements in BLWT
By using 3D print technology, model of the structure in scale 1:300 with rough façade was created. Experimental measurements were performed in Boundary Layer Wind Tunnel of Slovak University of Technology in Bratislava see figure 8. Wind tunnel is designed with open circuit scheme (see Hubova et al. [3], [6]) with overall length 26.3 m and two operation sectors of cross-section 2.6 x 1.6 m and with adjustable ceiling. The turbulent wind flow is created in rear operating space. In our case, the roughness of floor was created by plastic film FASTRADE 20 and 150 mm barrier. From the evaluation of the vertical mean velocity profiles, it seems that this modification of floor of tunnel is in the match with terrain category III - IV (closely to IV) with roughness length $z_0 = 0.7(m)$, according to ACSE [4], EN 1991-1-4 [5] and Wieringa [7].

![Figure 8. Description and view on Boundary layer wind tunnel](image)

To measure the mean wind speed in the open top of building, we use the Irwin omni directional probes, see figure 9, which were evaluated using a digital pressure scanner DSA 3217 from Scanivalve and we obtain average wind speeds and standard deviations. The model was placed on a rotating table
(figure 10), which was used for simulation of various wind directions with step of rotation $15^\circ$. Pressure taps were placed on the façade in height 92.7m (30.9 cm on model) and 97.8 m (32.6 cm on model) to measure external wind pressure.

Figure 9. View on the position of Irwin probes

Figure 10. 3D model in BLWT for wind $90^\circ$

Reference wind speeds (6.95, 8.68 and 10.98 m/s) were selected with regard to ACSE [4] to fulfill flow similarity of prototype and model and were measured by ALMEMO probe in height $H=33.6$ cm - 100.8 m on real structure. Reynolds numbers for 3 reference velocities reached values higher than recommended value $10^4$. Average temperature during tests was 15.4°C, barometric pressure was 100 920 Pa and air density was 1.215 kg/m$^3$.

Models of high-rise objects (in the middle) with surrounding buildings located in the wind tunnel during experiments, when the northwest wind is flowing, can be seen in figure 11.

Figure 11. View of the models placed on the turntable during the experimental measurements

5. Results and discussions

The aim of the work was to use the space of two technical floors for the possible placement of small vertical wind turbines with a vertical axis and to achieve sufficient acceleration of the wind flow at the top of the building. Figure 12 shows the average wind velocity at the site of the individual turbines obtained by experimental measurement for the most common wind directions in locality. As shown in the figure, at least 2 turbines always work at maximum power.

Due to the significant increase in small vortices at the entrance, that have been experimentally and numerically detected, it is necessary to insert a rectifying grid that will optimize the wind flow in inlet region.
We compared the results of numerical simulation and experimental measurements for different wind directions and the results were in good agreement. Comparison of wind acceleration in space of wind turbines for different wind directions is shown in figure 13.

![Mean wind velocity at the position of Irwin probe](image)

**Figure 12.** Mean wind velocity at the position of small wind turbines for different predominant wind directions and $v_{ref} = 10.98\, \text{m/s}$

![Comparison of wind acceleration](image)

**Figure 13.** Comparison of wind acceleration for individual left, middle and right wind turbine positions for different wind directions

Comparison of external wind pressure at the top of a building near the modelled area for wind turbines is seen in Figure 14.
6. Conclusions

We monitored the distribution of wind velocity on the high-rise building with surrounding area (in scale 1:300). The orientation of the building in relation to the prevailing wind directions plays an important role in the wind energy utilization.

The wind speed distribution results obtained by CFD and experimental measurements in BLWT allow us to select a suitable VAWT type, see Battisti et al. [1] in this area and height above ground in Bratislava city centre. In accordance with power curves for different types of small wind turbines it appears that three-bladed configuration H-rotor DU06W200 and 2-bladed H rotor NACA0018 will have maximal annual energy output.

The wind energy potential is based on average local wind velocities as well as the IRWES wind energy characteristics. In Bratislava city centre is mean wind velocity at height 10m in Northwest wind direction 4 - 5 m/s. The gradient of average wind speed at 100 meters will increase this value to around 8 m/s. New design of the entrance and exit areas at the top of the object will ensure sufficient wind acceleration for other wind directions. Annual energy output for wind velocity higher than 8m/s for the considered 3 turbines and 3 buildings should give 70 MWh/year.

Acknowledgment(s)
The presented results were achieved under sponsorship of the Grand Agency VEGA of the Slovak Republic (grant. reg. 01/0265/16) and with the support of the TU1304 COST action “WINERCOST”.

References
[1] L. Battisti, A. Brighenti, E. Benini, M.R. Castelli, “Analysis of Different Blade Architectures on small VAWT Performance”, In: Journal of Physics: Conference Series 753 IOP Publishing 1088/1742-6596/753/6/062009, pp. 1-8, 2016.
[2] A. B. Suma, R. M. Ferraro, B. Dano, and S. P. G. Moonen, “Integrated Roof Wind Energy System” In 2nd European Energy Conference, Maastricht, The Nederlandes. (2012).
[3] O. Hubova, P. Lobotka, “The multipurpose wind tunnel STU”, In Civil and Environmental Engineering Scientific - Technical Journal. ISSN 1336-5835, EV 3293/09, Volume Volume 10th, Issue 1/2014, pp. 2 - 9.
[4] ACSE Manuals and Reports on Engineering Practice, no.67. Wind Tunnel studies of buildings and structures Aerospace Division of the American Society of Civil Engineers, ISBN 0-7844-0319-8, 1999.
[5] EN 1991-1-4 Eurocode 1: Actions on structures-Part 1-4: General actions-Wind actions, 2005
[6] O. Hubova, P. Lobotka, “The Natural Wind Simulations in the BLWT STU Wind Tunnel“. Vienna: TGM - Federal Institute of Technology, 2014, In: ATF 3rd Conference on Building Physics and Applied Technology in Architecture and Building Structures. E-Book of reviewed papers Vienna, Austria, 6.-7.5.2014, pp.78-84 ISBN 978-3-200-03644-4, 2014.

[7] J. Wieringa, “Representative Roughness Parameters for Homogeneous Terrain”, In: Boundary Layer Meteorology, 63, 323-363, 1993.

[8] J. Franke, et al., ”Best practice guideline for the CFD simulation of flows in the urban environment” COST Office Brussels, 2007.

[9] Y. Tominaga, “AIJ guidelines for practical applications of CFD to pedestrian wind environment around buildings”, J Wind Eng and Aerod. 2008. 96, (10-11) 1749-1761.

[10] ANSYS Inc., ANSYS Fluent User’s Guide, Release 18.1, 2017

[11] D.C. Wilcox, “Multiscale model for turbulent flows”. In: AIAA 24th Aerospace Sciences Meeting. American Institute of Aeronautics and Astronautics, ISBN: 978-1-84821-001-1, 1986.