Numerical Investigation of an Atypical Building in Turbulent Wind Flow

Olga Hubova, Marek Macak
Slovak University of Technology, Faculty of Civil Engineering, Radlinskeho 11, 810 05 Bratislava, Slovakia

olga.hubova@stuba.sk

Abstract. The article deals with the numerical investigation of wind flow and wind pressure distribution on an atypical building and influence of wind direction on the external wind pressure coefficients. The analysis and computer simulation have been solved due to the collapse of the bottom of the overhanging parts that have been damaged by a strong wind. In the first phase, the selected building and its surroundings were investigated using CFD simulation. We chose the finite volume method implemented into the program ANSYS Fluent, which offers several turbulence models. RANS k-ε model was used for our solution for near-wall treatment; the standard wall functions by Launder and Spalding were used. Additional inputs for the k-ε model are equations for turbulent kinetic energy k and turbulence dissipation rate ε. Size of the computational domain was used according to the recommended value of block ratio of 3%. The wind pressures obtained at the bottom of the ceiling were comparable with the Eurocode values and were not significantly higher. In the second phase, we tested the pressure distribution at the bottom of the suspended ceiling, assuming that the wind could enter the interspace of the ceiling. In the north wind direction, the streets around the building accelerate the wind flow, the wind runs down on the building's façade and gets in the bottom of the suspended ceiling through the openings and tears off the soffit because the resulting pressures are significantly higher than the Eurocode values for roofs.

1. Introduction
In recent years, strong wind storms have appeared in Europe, accompanied by high wind speeds. In the case of a windstorm where the wind speed reaches values greater than 32.7 m/s (118km/h), we are talking about orkan. Western and Central Europe have been hit by several orkans, in 2007 the Orkan Kyrill, where the gust wind speed reached up to 250 km/h, in 2008 Emma with the maximum wind speed of 236 km/h, in 2010 Orkan Xynthia with a wind speed of 238 km/h. In 2020 it was Orkan Sabine where gust wind speed reached 202 km/h. These high wind speeds have caused enormous damages to the building structures in several countries. Problems arose on the roofs, but also in the corners of buildings and ceilings where high suction values were generated.

The distribution of wind pressure on structures had been tested in the past by several authors [1-3]. The results for simple shapes and regular types of structures are processed in Eurocode 1991-1-4. Irregular and atypical structures are the subject of numerical and experimental research in order to draw attention to the extremes that may arise when the wind is flowing around a structure having an
irregular shape. Nowadays, modern numerical CFD methods are used to solve the problems of fluid flow.

2. Numerical simulation of wind effects on an atypical object

The objective of the numerical simulation was to define critical suction values at the bottom of the cantilever part and wind pressures distribution on the object (see figure 1) for different wind directions. For the analysis of our problem, we chose the finite volume method implemented into program ANSYS Fluent [4, 5] which offers several turbulence models.

![Figure 1. View of an atypical building with damaged cantilevered part](image)

The problem presented in this article is a preliminary modeling of the wind flow at the site of the atypical building in Slovakia using FLUENT ANSYS program, which outlines the adverse effects of the wind flow on the building.

2.1. Numerical model

Computer fluid simulation for the building in scale 1:1 was prepared. For the solution of the 3D steady RANS equations with the standard \( k-\varepsilon \) model [6], we used CFD code ANSYS Fluent, which solves Navier Stokes equations along with the added two equations. Turbulent kinetic energy is solved by the equation:

\[
\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_j}(\rho k u_j) = \frac{\partial}{\partial x_j} \left[ \left( \frac{\mu}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right] + G_k + G_S - \rho \varepsilon - Y + S_k
\]  

Dissipation of turbulent kinetic energy \( \varepsilon \) is calculated by the equation:

\[
\frac{\partial}{\partial t}(\rho \varepsilon) + \frac{\partial}{\partial x_j}(\rho \varepsilon u_j) = \frac{\partial}{\partial x_j} \left[ \left( \frac{\mu}{\sigma_\varepsilon} \right) \frac{\partial \varepsilon}{\partial x_j} \right] + \rho C_{1\varepsilon} S_k - \rho C_{2\varepsilon} \frac{\varepsilon^2}{k + \sqrt{\varepsilon}} + C_{3\varepsilon} \frac{\varepsilon}{k} C_{4\varepsilon} C_{5\varepsilon} S_k + S_\varepsilon
\]

This turbulent model is characterized by high accuracy near the walls, while the accuracy in the open terrain is average.

2.2. Computational domain

Size of the computational domain was \( 1.5 \times 1 \times 0.2 \) km\(^3\) (l×w×h) according to the recommended maximum value of block ratio 3% [7, 8]. The distance from the triangle object to the inlet, sides and top of the domain is at least five times the height of the building, and the distance from the outlet is at
least eleven times the height, according to the guidelines by [7, 8]. The computational domain is illustrated in figure 2.

![Figure 2. Illustration of the computational domain in ANSYS Fluent a/ and detail b/](image1)

2.3. Computational grid (mesh)
We created a mesh with the size function set on distance using the polyhedral elements. The element size on the surface of the investigated building was 0.2 m with soft behavior using curvature function and the inflation for these surfaces was applied with 5 layers with the height of the first layer 0.05 m. The element size on surfaces of the bottom of the cantilevered part was 0.05 m. For the surrounding buildings, the size of the elements was 3 m. The element size on another surface was 12 m with soft behavior using distance and curvature function. On the bottom boundary layer, the inflation with 5 layers with the height of the first layer 0.5 m according to [7-9] was applied. There were generated 2 491 863 nodes and 2 267 520 elements. A computational grid is illustrated in figure 3.

![Figure 3. Illustration of the computational grid of the surrounding a/ and detail of the building b/](image2)

2.4. Boundary conditions
The inlet boundary conditions of the domain are defined by the vertical profiles

\[ v(z) = \frac{v^*}{\kappa} \ln \frac{z + z_0}{z_0}, \quad v^* = \frac{v_{ref} \kappa}{\ln \frac{z_{ref} + z_0}{z_0}}, \]

where \( v(z) \) is mean wind velocity at height \( z \), \( v^* \) is shear velocity, \( z_0 \) is aerodynamic roughness height (\( z_0 = 0.3 \) terrain cat. III and \( z_0 = 0.05 \) terrain cat. II), \( \kappa \) is von Karman constant (\( \kappa = 0.42 \)). The reference wind speeds are known for the reference height \( z_{ref} = 23.975 \) m and terrains II and III.
\( v_{\text{ref}} = 24.242 \text{ m/s} \) for terrain III and \( v_{\text{ref}} = 30.358 \text{ m/s} \) for terrain II.

The turbulence at the entrance to the computational area is modeled using the relationships:

\[
k = \frac{u''}{\sqrt{C_p}} \quad \text{and} \quad \varepsilon(z) = \frac{u''}{\kappa(z + z_v)},
\]

\( C_p = 0.01 \) is a model constant. These boundary conditions create the wind and turbulence profile that is identical to the EN 1991-1-4.

The outlet boundary is defined as pressure outflow and the side and upper boundary as zero gradient (symmetry).

2.5. Achieved results

Due to a large number of simulations, we summarize obtained results in the table illustrating the most critical directions. The external wind pressure coefficients for the building are shown in Table 1 for all investigated wind directions. The highest suction values occurred on the roof of the building, but the local values are not higher than those given in the EN standard for roofs. The maximum wind pressure coefficients exceeded the local pressure values \( c_{pe,1} = 1 \) from EN standard (see figure 4).

| Building | Wind direction | Terrain category | angle [°] | min \( c_{pe} \) | min \( w_e \) | Height [m] | Max \( c_{pe} \) | max \( w_e \) | Height [m] | \( v_{\text{ref}} \) [m/s] |
|----------|----------------|----------------|----------|----------------|----------------|------------|----------------|----------------|------------|----------------|---------------|
|          |                | 3              | 0        | -1.215         | -446.275       | 23.975     | 1.1259        | 413.539       | 26.077     | 24.242         |
|          |                | 3              | 11.25    | -1.422         | -522.132       | 23.975     | 1.1488        | 421.95        | 25.879     | 24.242         |
|          |                | 3              | 45       | -0.7043        | -258.687       | 23.975     | 0.5539        | 203.458       | 21.926     | 24.242         |
|          |                | 3              | 90       | -0.7621        | -279.918       | 23.975     | 0.5043        | 185.214       | 23.309     | 24.242         |
|          |                | 3              | 112.5    | -1.8091        | -664.477       | 23.975     | 1.1079        | 406.927       | 19.949     | 24.242         |
|          |                | 2              | 135      | -1.7194        | -990.387       | 23.975     | 0.7749        | 446.348       | 15.996     | 30.358         |
|          |                | 2              | 180      | -0.8006        | -461.162       | 15.200     | 0.7934        | 457.018       | 14.613     | 30.358         |
|          |                | 2              | 225      | -1.4269        | -821.88        | 15.200     | 0.7919        | 456.12        | 12.834     | 30.358         |
|          |                | 3              | 270      | -1.2166        | -446.85        | 27.465     | 0.7836        | 287.803       | 26.867     | 24.242         |
|          |                | 3              | 292.5    | -2.0077        | -737.419       | 24.855     | 0.8099        | 297.464       | 21.135     | 24.242         |
|          |                | 3              | 315      | -1.5173        | -557.305       | 15.900     | 0.97797       | 359.207       | 21.728     | 24.242         |
|          |                | 3              | 337.5    | -0.9930        | -364.732       | 15.900     | 0.6121        | 224.815       | 26.472     | 24.242         |
|          |                | 3              | 348.75   | -0.87915       | -322.908       | 24.855     | 0.7518        | 276.121       | 26.472     | 24.242         |
|          |                | Extreme values |         | -2.0077        | -990.387       | 1.1488     | 457.018       |               |            |                |

Due to the damage and tearing off the soffit of the console of the building, we evaluated the wind pressure coefficients in this area. We assumed that the cladding of the cantilever part would be bypassed by the wind. The external wind pressure coefficients of the ceiling are shown in figure 5 and Table 2. The value of the largest suction did not exceed the EN standard value of the local suction in the corners \( c_{pe,1} = -1.4 \).
Figure 4. External wind pressure coefficients for wind currents from the north (0°)

Figure 5. External wind pressure coefficients on the soffit of the cantilever part of the building

The most unfavorable values occurred in the north and the south-east wind directions, where the values are higher, because the wind has a higher speed as it crosses the terrain category II, which is a smoother terrain. Suction appears in the corners and in the cantilever part from the columns to the free end. The values of the external suction are not significant, provided that the wind does not enter the suspended soffit, see figure 5.
Table 2. External wind pressure and pressure coefficients in the soffit

| Terrain category | Wind direction | Soffit at height 6.72m |
|------------------|----------------|------------------------|
|                  | angle          | min $c_{pe}$ | min we  | max $c_{pe}$ | max we  | $v_{ref}$ |
|                  | [°]            | [Pa]        | [Pa]    | [Pa]        | [m/s]   |          |
| 3                | 0              | -0.609      | -223.839| 0.7616      | 279.748| 24.242   |
| 3                | 11.25          | -0.3621     | -132.994| 0.6591      | 242.067| 24.242   |
| 3                | 45             | -0.1964     | -72.1206| 0.4011      | 147.329| 24.242   |
| 3                | 90             | -0.3239     | -118.987| 0.1827      | 67.0894| 24.242   |
| 3                | 112.5          | -0.7793     | -286.242| 0.3279      | 120.422| 24.242   |
| 2                | 135            | -1.3428     | -773.428| 0.061       | 35.119 | 30.358   |
| 2                | 180            | -0.3028     | -174.432| 0.0168      | 9.6752 | 30.358   |
| 2                | 225            | -0.1783     | -102.717| 0.0887      | 51.0669| 30.358   |
| 3                | 270            | -0.0356     | -13.098 | 0.0841      | 30.8955| 24.242   |
| 3                | 292.5          | -0.5551     | -203.878| 0.2566      | 94.2402| 24.242   |
| 3                | 315            | -0.5091     | -186.996| 0.593346    | 217.934| 24.242   |
| 3                | 337.5          | -0.2183     | -80.1791| 0.093       | 34.1491| 24.242   |
| 3                | 348.75         | -0.4215     | -154.799| 0.201       | 73.7902| 24.242   |

Extreme values

-1.3428 -773.428 0.761641 279.748

We have also solved the problem assuming that the wind can get through the holes around the corners, joints and through the cladding to the interior of the ceiling. Assuming that the wind can penetrate the soffit from the north, which can occur if the cladding and openings are permeable, and also when it is caused by the self-weight of the soffit, the resulting wind pressure coefficients increase, see figure 6.

Figure 6. External wind pressure coefficients in the case when the wind flow enters the soffit
The resulting wind pressure coefficients, as shown in figure 6, can reach value $c_{pe,1} = -2.8$. The streamlines around the object in the north wind are shown in figures 7 and 8.

![Figure 7. Streamlines around an atypical building](image7)

![Figure 8. Close-up view of the streamlines from the side](image8)

3. Results and discussions
The aim of the computer simulation of the wind flow was to determine the probable cause of the building's soffit tear off during a wind storm hitting the building from the north direction. The CFD simulation of wind flow around an atypical building showed only slightly higher wind pressure values on the building. The biggest suction occurred on the roof and roof sections, but did not exceed the standard values.
In a model situation where the wind could get in the soffit, the resulting values of the external wind suction coefficient were 4 times higher than if the wind only bypassed the soffit. From the achieved results and from the comparison of two different situations, it is clear that the entry of wind into the soffit should be prevented.

4. Conclusions
The result of the numerical analysis is that the soffit should be designed for local suction values similar to those of roof sections, where the external wind pressure coefficients can reach values of -2.8.

Acknowledgment(s)
Presented results have been arranged due to the research supported by the Slovak Scientific Grant Agency, projects VEGA No. 1/0412/18 and No. 1/0453/20.

References
[1] A.G. Davenport, “New approaches to the design of structures against wind action,” Canadian Structural Engng. Conference, 1968.
[2] J.E. Cermak, “Separation-induced pressure fluctuations on buildings,” Paper at US-Japan Seminar Winds Loads on Structures, University of Hawaii, 1970.
[3] G. Solari, “Gust buffet ing. I: Peak wind velocity and equivalent pressure,” J. Struct. Engng., ASCE, 119, 2 365-382, 1993.
[4] ANSYS Inc., ANSYS Fluent Theory Guide User’s Guide, Release 18.1, 2017.
[5] ANSYS Inc., ANSYS Fluent User’s Guide, Release 18.1, 2017.
[6] 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.
[7] Franke J. et al., Best practice guideline for the CFD simulation of flows in the urban environment, COST Office Brussels, 2007.
[8] Y. Tominaga, AIJ guidelines for practical applications of CFD to pedestrian wind environment around buildings, J Wind Eng Ind Aerod 2008, 96, (10-11) 1749-1761, 2008.
[9] B. Blocken, T. Statopoulos, and J. Carmeliet, CFD simulation of the atmospheric boundary layer–wall function problems. Atmospheric Environment 41(2):238-252. Elsevier, 2007.