Iacob-Liviu SCURTU, Alexandra-Daniela SCURTU

AERODYNAMICS EVALUATION OF HIGH-RISE BUILDINGS

Abstract: *Due to the increasing population density in metropolitan areas, architects have focused on high-rise building designs that combine architectural considerations with functional requirements. The optimal design of the building shape in the case of wind interaction can be determined using the CFD evaluation method. This study uses the CFD analysis method to analyse the airflow around a new high-rise building. The first part of the paper presents a comprehensive overview of the state of the art regarding the building shapes used and the main simulation method. In the second part of the paper, the buildings model is created using dedicated CAD software. Two simulation cases are being prepared with two wind flow situations at 15 m/s velocity. In the third part of the paper, the results of CFD analysis of the buildings group model are presented. The conclusions of this evaluation are shown in the last part of the paper.*

Keywords: *buildings aerodynamics, CFD simulation, high-rise buildings, wind streams*

1. INTRODUCTION

The expansion of real estate and the maximum utilization of building land are causing discomfort among residents in large cities. One of the common situations is the construction of buildings that are much higher than those in the surrounding area and which are deprived of sunlight, as well as the additional pressure exerted on their surface when the wind is stronger.

The geometry of the building has a direct effect on the abnormal wind flow and leads to greater pressure on the building surface or the surrounding buildings. A building is considered tall if it is more than 60 meters high. Considering the impact that such a structure has on a community, research and studies are conducted on the airflow around the building [1].

It is known from the beginning that wind power is variable, and an aerodynamic study is difficult to achieve with high accuracy. Aerodynamic shapes used in the construction of high-rise buildings can reduce the vibration that can occur in the building structure and the vortices generated by the wind. For example, if the building has a cylindrical shape and the wind is from the back of the building with a Reynolds number in a range of 100 to 10^5, the vortices are called "von Karman vortex street" [2]. This phenomenon usually occurs in buildings that have a uniform volume. One way to reduce these vortices is to build tall buildings with external shapes that have a different cross-section over one entire height. The current tallest building in the world, the Burj Kalifa (828 m) in Dubai, United Arab Emirates, is designed to reduce the formation of the air vortex in strong winds [3]. An important review of the buildings aerodynamics was published by Aiman H.H. Al-Masoodi, who describes in detail how air flows around high-rise buildings, showing the studies on the basic external building shapes of skyscrapers and the optimal shapes, derived from the basic shapes [4].

The latest method for predicting air flows around buildings is the CFD method (Computational Fluid Dynamics), which is an iterative method of calculation and simulation using cartesian grid FEM. Devesh Kasana investigates the air pressure resulting from airflow around the buildings using the CFD method integrated into Ansys CFX software [5]. Yukio Tamura presents a study showing the variation of the external building shapes that can be generated from the basic models. Modified shapes are created by weaving or rounding the edges of the base volume, twisting the basic pattern or reducing the surface area. Removing material from the upper side of the model also influences a more optimal airflow around high-rise buildings [6,7]. To ease the work, in the last time, many CADS software integrated the CFD solver to help the user to optimize the final shape of the model.

This study summarizes the airflow analysis of the 95 m high-rise City Tower building from Cluj-Napoca, with nearby buildings, which have a much lower height.

2. SETUP EVALUATION

2.1 CAD configuration

This study shows the wind flow evaluation around a group of buildings from Cluj-Napoca. The CAD model is composed of seven buildings presented in Figure 1.

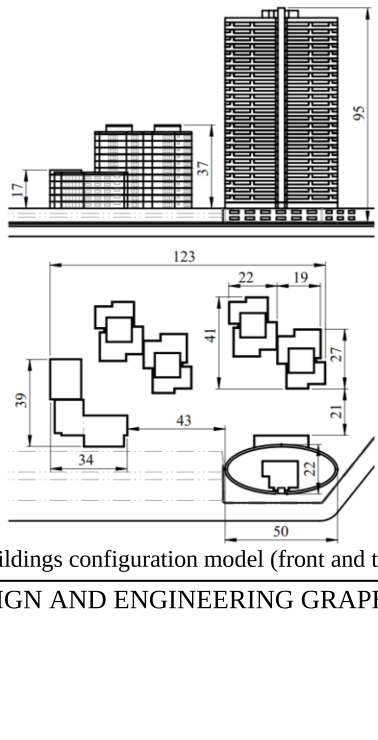


Figure 1 Buildings configuration model (front and top view).

The building dimensions and configuration are chosen from the Google Earth application, respecting several geometry details in the modelling phase. Units for all dimensions are given in meters. The CAD model of the building configurations is done by using SolidWorks software. To evaluate the pressure impact on the small buildings surface it is necessary to create a target surface to collect the pressure values. To establish the pressure effect on the buildings surface, these are named as can be observed on the CAD model from Figure 2.

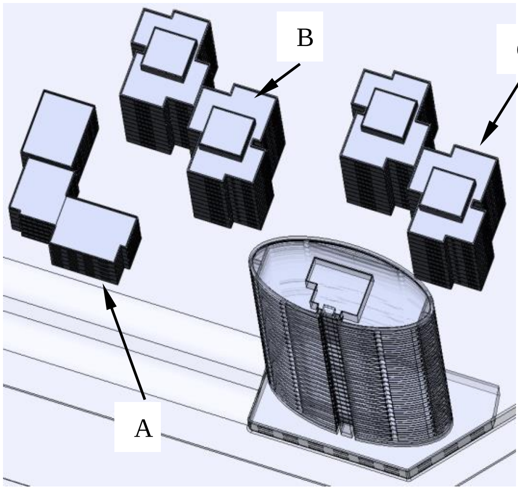


Figure 2 Buildings CAD model.

The target surfaces chosen are presented in Figure 3, for two simulation cases. Because the C building is the same as the B, this is not presented in the picture. In first simulation case are chosen the surface: A1.1, A1.2, A1.3, B1.1, B1.2, B1.3, C1.1, C1.2, C1.3 and in second case: A2.1, A2.2, A2.3, B2.1, B2.2, B2.3, C2.1, C2.2, C2.3.

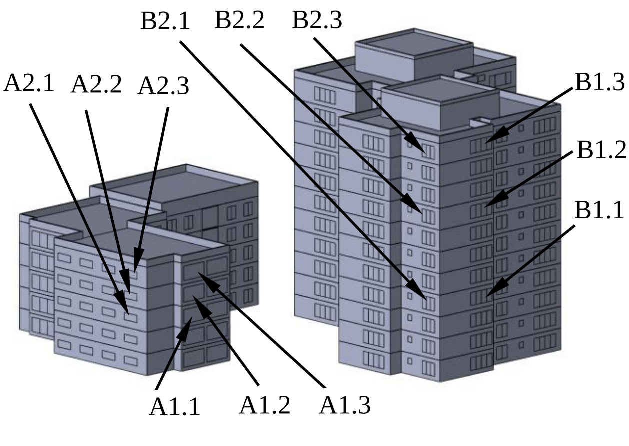


Figure 3 Target surfaces to pressure effect evaluation.

It can be observed that the surface is chosen starting at the last three floors of A building and the fourth floor at the B and C buildings.

2.2 CFD setup

This section presents the methodology to create the CFD analysis of the modelled group of buildings. The simulation analysis is done using a CFD solver from SolidWorks software. Knowing that the natural wind flows in a velocity range of 1 to 50 m/s, in this study a wind flow velocity V=15 m/s is applied for both simulation cases. This wind flow velocity is chosen because the buildings are placed in an open area, without landforms that could significantly reduce the wind flow. It is considered that the air density is ρ=1.2041[kg/m^3] at temperature T=20°C. Before starting the CFD analysis it is necessary to establish whether the airflow behaviour is laminar or turbulent. By calculating the Reynolds number, it can be established the flow regime or whether the "von Karman vortex street" may occur. The expression 1 represents the formula used to calculate the Reynolds number:

$$R_e = \frac{\rho \cdot V \cdot L}{\mu}. \qquad (1)$$

where, L= 122.5 m, represent the building length and μ = 1.802 · 10^5 kg/m·s. is the dynamic air viscosity. The value of the Reynolds number is 1.23·10^8, a fact which shows that no "von Karman vortex street" appears.

The first simulation case is prepared as can be shown in Figure 4, the wind flows parallel with the vehicle's road. The CFD domain volume is chosen to view the airflow behaviour around the building surface. The parameters of the CFD domain are presented in Figure 4.

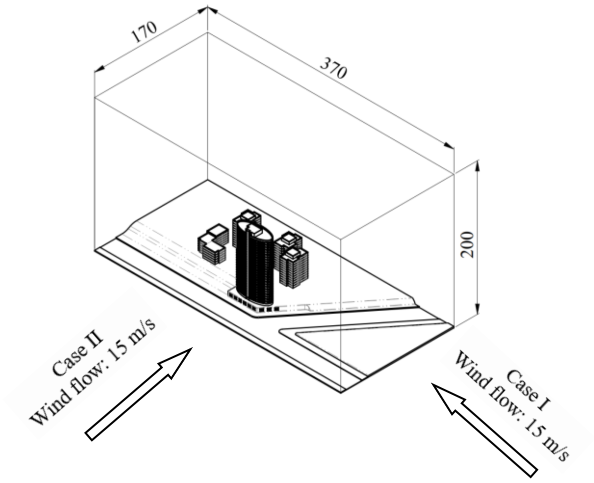


Figure 4 CFD volume dimensions.

The buildings model is represented on a real scale. The CFD volume is divided into 458181 fluid cells, including 85149 fluid cells that are in contact with a solid.

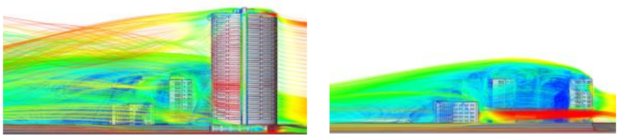
3. RESULTS AND DISCUSSIONS

In this section, the values obtained for the airflow pressure and the air velocity are presented. Each analysis case contains two CFD simulations: first without high building and second with high building. Table 1 presents the relative pressure on the windows surface for the first case, when the airflow is parallel with the road. Analysing the results, it can be observed that the pressure on the first C building is higher in the case with high building.

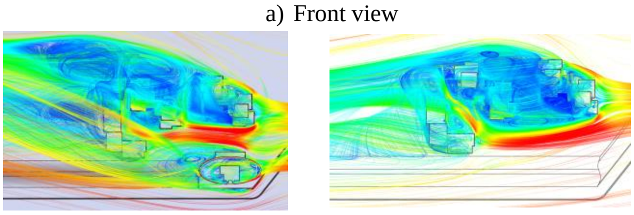
Table 1 Case 1 – Pressure distribution-averaged value [Pa].

Case I - Simulation 1- airflows along on roadside					
Pressure on perpendicular windows with roadside – with high building					
A		**B**		**C**	
A1.1	101382	B1.1	101317	C1.1	101472
A1.2	101375	B1.2	101316	C1.2	101467
A1.3	101363	B1.3	101323	C1.3	101433
Pressure on parallel windows with roadside – with high building					
A		**B**		**C**	
A2.1	101282	B2.1	101335	C2.1	101249
A2.2	101281	B2.2	101337	C2.2	101250
A2.3	101282	B2.3	101324	C2.3	101257
Pressure on perpendicular windows with roadside – without high building					
A		**B**		**C**	
A1.1	101433	B1.1	101279	C1.1	101467
A1.2	101467	B1.2	101277	C1.2	101460
A1.3	101457	B1.3	101281	C1.3	101431
Pressure on parallel windows with roadside – without high building					
A		**B**		**C**	
A2.1	101548	B2.1	101548	C2.1	101221
A2.2	101534	B2.2	101530	C2.2	101226
A2.3	101466	B2.3	101453	C2.3	101234

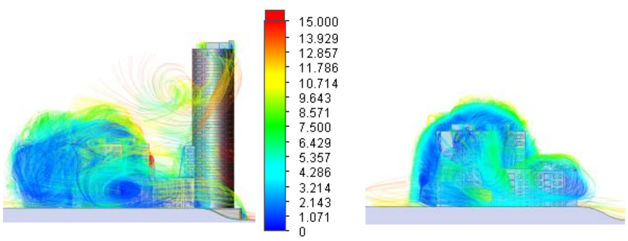
Figure 5 presents the comparative images of case I, showing the airflow velocity behaviour around the buildings group in two configuration: first configuration with higth-rise building and second configuration without higth-rise building.



a) Front view



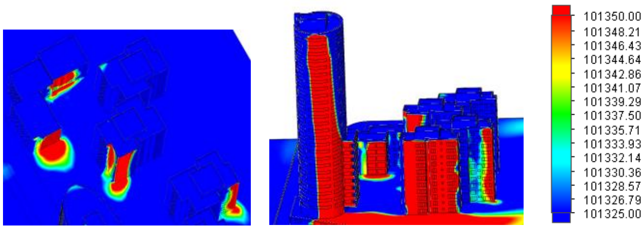
b) Top view



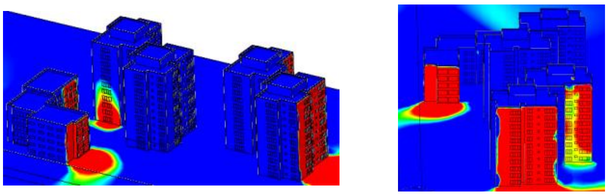
a) Side view

Figure 5 Comparative wind velocity distribution-case I [m/s].

Analysing the airflow it can be observed that the lower velocity area is located in the rear side of the buildings in case of hight-rise builildings, the air flux is orientented among the lower buildings at more higher velocity resulting the lower comfort for residents from adjacent buildings. The airflow pressure distribution is represented in Figure 6 for the first case. In figure 6a and 6b are presented two pictures for each simulation that shows high pressure distribution on the external buildings surface.



a) Airflow pressure with high building



b) Airflow pressure without high building

Figure 6. Wind pressure distribution- case I - [Pa].

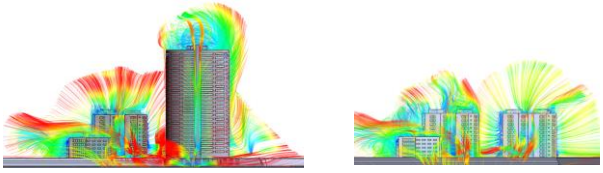
Anlysing the images from Figure 6 it can be observed that on the right side of the C building the pressure is lower when the high building is in simululation and on the second body of the B building the pressure is higer in simulation without a higth-rise building.

Table 2

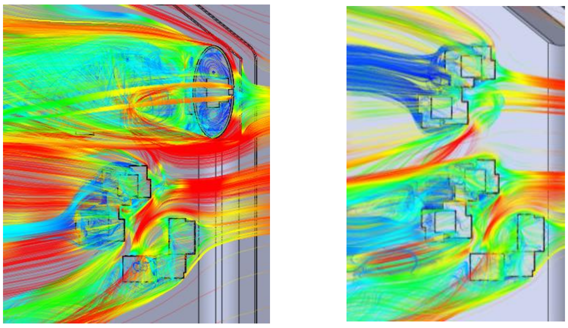
Case 2 – Pressure on target surface-averaged value [Pa].

Case II - Simulation 1- airflows perpendicular on roadside					
Pressure on perpendicular windows with roadside – with high building					
A		**B**		**C**	
A1.1	101383	B1.1	101319	C1.1	101299
A1.2	101375	B1.2	101321	C1.2	101300
A1.3	101367	B1.3	101320	C1.3	101303
Pressure on parallel windows with roadside – with high building					
A		**B**		**C**	
A2.1	101570	B2.1	101573	C2.1	101285
A2.2	101570	B2.2	101552	C2.2	101284
A2.3	101562	B2.3	101517	C2.3	101286
Pressure on perpendicular windows with roadside – without high building					
A		**B**		**C**	
A1.1	101384	B1.1	101307	C1.1	101314
A1.2	101367	B1.2	101294	C1.2	101313
A1.3	101366	B1.3	101301	C1.3	101314
Pressure on parallel windows with roadside – without high building					
A		**B**		**C**	
A2.1	101518	B2.1	101548	C2.1	101467
A2.2	101534	B2.2	101530	C2.2	101473
A2.3	101521	B2.3	101466	C2.3	101464

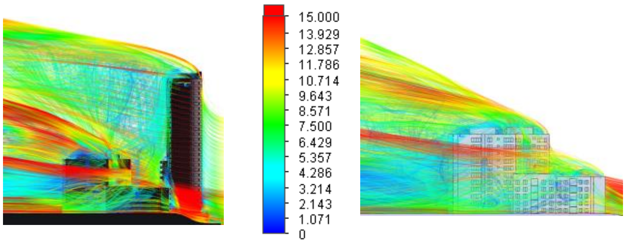
In the second case, when the wind flow on a perpendicular direction with a roadside, the wind velocity is smooth when the building group is without a high building. When the high building is on the group, the airflow velocity has a turbulent behaviour between buildings and on the rear side of the building group, as can be observed in Figure 7.



c) Front view



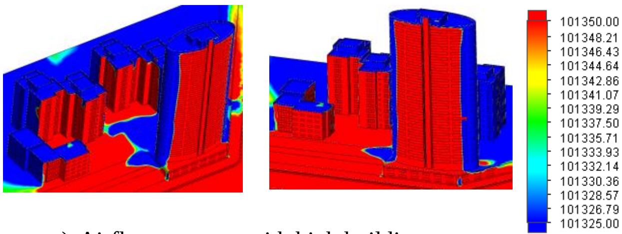
d) Top view



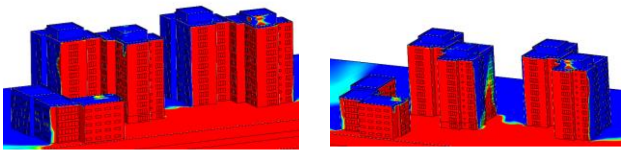
b) Side view

Figure 7 Comparative wind velocity distribution-case II [m/s].

In the second simulation case, the air pressure on the buildings surface is higher in the case with the higher building. The high building has a shield effect for the C group of buildings.



a) Airflow pressure with high building



b) Airflow pressure without high building

Figure 8. Wind pressure distribution- case II - [Pa].

4. CONCLUSIONS

In this study the CFD simulation evaluation of high-rise buildings are presented. The effects at an air speed of 15 m/s do not seem to alter the quality of life for residents in the vicinity of the large block. One problem is that the high building blocks natural light at some times of the day. Aerodynamic analysis of buildings is recommended to be done from the project phase, to find the possible problems that could have negative consequences on the resident's comfort.

As a drawback, the obtained results are not exact with reality because the orientation of the airflow cannot be exactly as defined in cases. Many parts that appear could not be represented, such as: terrain geometry, cars that are on the roadside, trees, etc.

REFERENCES

[1] Kasana, D., Tayal, D., Choudhary, D., Raj, R., Meena, R. K., & Anbukumar, S. (2022). Evaluation of aerodynamic effects on a tall building with various cross-section shapes having equal area. *Forces in Mechanics*, *9*, 100134.

[2] Irwin, P. A. (2010). Vortices and tall buildings: A recipe for resonance. Physics Today, 63(9), 68-69.

[3] J.E. Cooper, Aeroelastic response, Editor(s): S. Braun, Encyclopedia of Vibration, Elsevier, 2001, Pages 87-97, ISBN 9780122270857, https://doi.org/10.1006/rwvb.2001.0125.

[4] Al-masoodi, A. H., Alkhatib, F. H., Shafiq, N., & Wahab, M. M. A. (2022, May). The Aerodynamic Performance of Tall Buildings by Utilizing Aerodynamic Modifications-A Review Study. In *IOP Conference Series: Earth and Environmental Science* (Vol. 1022, No. 1, p. 012046). IOP Publishing.

[5] Kasana, D., Tayal, D., Choudhary, D., Raj, R., Meena, R. K., & Anbukumar, S. (2022). Evaluation of aerodynamic effects on a tall building with various cross-section shapes having equal area. *Forces in Mechanics*, *9*, 100134.

[6] Tamura, Y., Tanaka, H., Ohtake, K., Kim, Y. C., Yoshida, A., Bandi, E. K., ... & Yang, Q. (2015). Aerodynamic control of wind-induced vibrations and flow around super-tall buildings. In *Proceedings of the 6th International Conference Adv. Exp. Strutural Eng. International Work. Adv. Smart Mater, Smart Strutures Technol. Urbana-Champaign*.

[7] Kim, Y. C., Tamura, Y., Tanaka, H., Ohtake, K., Bandi, E. K., & Yoshida, A. (2014). Wind-induced responses of super-tall buildings with various atypical building shapes. *Journal of Wind Engineering and Industrial Aerodynamics*, *133*, 191-199.

Authors:

Lecturer PhD. Eng. Iacob-Liviu SCURTU, Technical University of Cluj-Napoca, Faculty of Automotive, Mechatronics and Mechanical Engineering, Department of Automotive Engineering and Transports,
E-mail: liviu.scurtu@auto.utcluj.ro

Researcher PhD. Eng. Alexandra-Daniela SCURTU, INCDO-INOE 2000 subsidiary Research Institute for Analytical Instrumentation ICIA, Cluj-Napoca, Corresponding author,
E-mail: daniela.scurtu@icia.ro