Wind flow around a church - case study

Paulina Jamińska-Gadomska[1]**, Tomasz Lipecki**[1] **and Jerzy Podgórski**[1]

[1] *Faculty of Civil Engineering and Architecture, Lublin University of Technology*
Nadbystrzycka 40, 20-618 Lublin, Poland
e-mail: p.jaminska@pollub.pl, t.lipecki @pollub.pl, j.podgorski @pollub.pl

Abstract

The abstract presents results of CFD analysis performed to check the influence of wind on a bell tower of a church. The geometry of the structure is quite complex therefore it is very hard to calculate the wind load only basing on codes recommendations. The modelled geometry contains whole structure of church including bell tower. CFD simulations have been performed for two inflow wind direction of opposite senses. This led to two cases of converging and diverging elements of the bell tower in relation to the wind flow. Such analysis was performed to check so called 'Venturi-effect' between the walls of the bell tower.

Keywords: CFD simulation, wind flow, Venturi-effect

1. Introduction

Wind load is one of the main loads which have to be taken into consideration in the designing process of a structure. It may have significant influence on a load bearing capacity of concerned building object. Eurocode [1] and other codes refers only to the common cases, such as rectangular, cylindrical or spherical shapes of structures. In reality a shape of the structure may be much more complicated what makes it hard to evaluate wind load properly. One solution to this problem is to conduct wind tunnel test on usually scaled down model and its surroundings. The other one is to perform Computational Fluid Dynamics (CFD) simulations. This abstract presents results of CFD simulation of a church building. The main focus was put on the bell tower of the church and the flow through its constricting walls.

2. Church – description of structure

The structure of discussed church (Fig. 1) can be generally divided in two parts – the main cylindrical structure with a sloping roof and the bell tower. The diameter of the cylindrical part is equal to 44 m, and the vertical dimension in the highest point is 15 m. The bell tower is created by two thin (0.4 m) walls coming from the ground as a broad structures to more slender ones, up to 38.5 m. The horizontal angle between these walls is equal to 100° and the passage between the walls of bell tower is 2.4 m.

Figure 1: View of the church and the bell tower

3. Computational grid

Computational grid was prepared in accordance to recommendations given in [2,3]. To speed up calculations the computational domain contained only half of the church geometry, since structure of the church was symmetrical and the flow direction was consistent with the church axis of symmetry. The computational domain length x width x height were following: 800 m x 125 m x 200 m. The inlet of the domain was located 200 m from the building. The outlet was set more than 500 m away from the building to enable undisturbed development of wake behind the structure.

4. CFD Simulations

Simulations were performed with use of commercial software ANSYS FLUENT. Atmospheric boundary layer was represented by a vertical wind speed profile which was set at the domain inlet. It was calculated according to the fallowing equations:

$$\overline{U}(z) = \frac{u_*}{\kappa} \ln\left(\frac{z+z_0}{z_0}\right) \tag{1}$$

$$u_* = \frac{u_{ref}\,\kappa}{\ln\left(\dfrac{z_{ref}}{z_0}\right)} \tag{2}$$

Friction velocity u_* was based on the reference wind speed u_{ref} = 10 m/s at the reference height z_{ref} = 10 m. The Aerodynamic roughness length z_0 was equal to 0.05 m and corresponded to the II[nd] terrain category [1]. Turbulent nature of wind was represented by turbulent kinetic energy $k(z)$ and turbulence dissipation rate $\varepsilon(z)$ which were defined as follows:

$$k(z) = \frac{u_*^2}{\sqrt{C_\mu}} \tag{3}$$

$$\varepsilon(z) = \frac{u_*^3}{\kappa(z+z_0)} \tag{4}$$

where: z – coordinate along the domain height, κ – von Karman constant (κ = 0.42), C_μ – model constant (C_μ = 0.09).

The boundary condition at the bottom of the domain was set to standard wall with no slip. The domain outlet was set to outflow.

The Reynolds-Averaged Navier-Stokes Equations were solved using RNG and realizable k-ε turbulence models. Pressure based solver and second-order discretization schemes were used in calculations.

5. Results and conclusions

The CFD simulations were performed for two wind directions of opposite senses. This led to two cases of converging and diverging walls of the bell tower in relation to the wind flow. The results obtained from CFD simulations gave data about wind patterns around the building. In both cases the maximum wind speed occurred at the top of the bell tower. Due to the diverging and converging wall arrangement the conducted study also checked if the shape of the bell tower walls can cause so called "Venturi effect". Similar issue discussing if converging walls can speed up the wind flow was undertaken by Blocken [4].

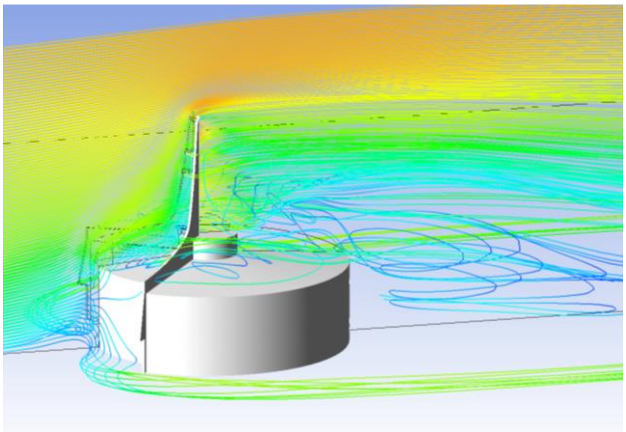

Figure 2: Flow around church

Since the church has complex geometry with few sharp edges, there are many points where the detachment of the boundary layer occurs and leads to vortex shedding. The highest values of the wind speed can be observed in the upper part of building, caused by sharp edges of the bell tower. Vortices can be seen in Figure 2 which shows wind pathlines for case of converging walls of the bell tower – wind blows from left to right of the model.

According to the conducted analyses there were no significant differences in maximum wind speed values between two considered cases. Figure 3 presents contours of the wind flow for two calculated cases at three heights: 37 m, 33 m, 25 m. The colour scale is the same in all figures. Blue colour correspond to zero whereas red to maximum magnitude of wind speed. Wind blows from left to right. As it can be seen there is no substantial speed up of the wind flow in the passage between bell tower walls. This can be due to the distance between walls which is big enough that it do not confine the flow. Another reason may be relative short walls which are easy for wind to pass by. Figure 4 presents contours of the wind flow at the roof height (15 m). The flow pattern clearly differs for diverging and converging cases, but the values of wind speed are quite similar.

a) b)

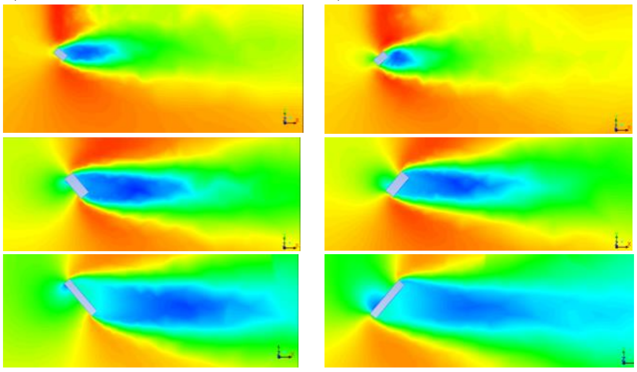

Figure 3: Wind flow patterns at height: 37 m, 33 m and 25 m: a) diverging, b) converging case

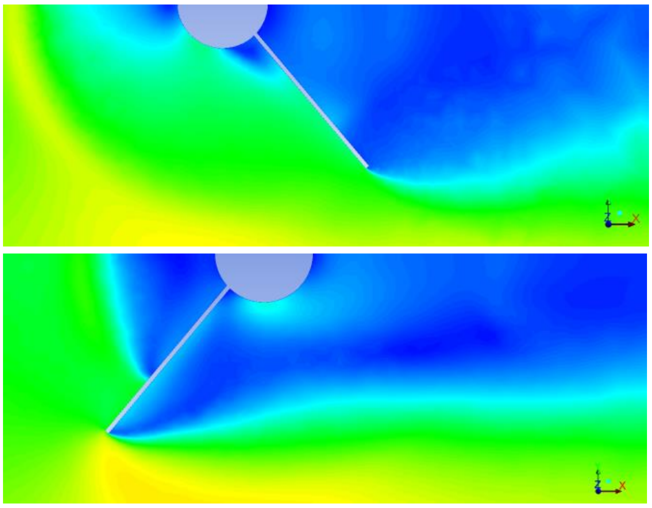

Figure 4: Wind flow patterns at height 15 m: top – diverging, bottom – converging case

References

[1] EN 1991-1-4:2008. Eurocode 1. General actions – Wind action.

[2] Franke, J., Hellsten, A., Schlünzen, H. and Carissimo, B., Best practice guideline for the CFD simulation of flows in the urban environment, COST Office, Brussels, 2007.

[3] Tominaga, Y., Mochida, A., Yoshie, R., Kataoka, H., Nozu, T., Yoshikawa, M. and Shirasawa, T., AIJ guidelines for practical applications of CFD to pedestrian wind environment around buildings, *J. Wind Eng. Ind. Aerodyn.*, 96, (10–11), pp. 1749–1761, 2008.

[4] Blocken, B., Moonen, P., Stathopoulos, T. and Carmeliet, J., Numerical study on the existence of the venturi effect in passages between perpendicular buildings, *J. Eng. Mech.*, 134, (12), pp. 1021–1028, 2008.