EFFECT OF THE BUILDING CROSS-SECTIONAL SHAPE ON THE COMFORT OF PEDESTRIANS

C. Stroumpouli [1], A. Camara [2], C. Jagadeesh [3]

[1,3]Department of Civil Engineering. City, University of London, London, EC1V 0HB, United Kingdom, Chrysanthi.Stroumpouli@city.ac.uk
[2]Department of Continuum Mechanics and Theory of Structures, Universidad Politecnica de Madrid, Profesor Aranguren s/n, 28040 Madrid, Spain.

SUMMARY:
There is growing interest in façades of buildings with moving screens that can adapt to different environmental conditions to optimise the energy usage and comfort. These façade elements can significantly affect the wind flow around the building, both at the local and the global levels. This research presents a programme of computational fluid dynamic (CFD) analysis to study the influence of the shape of the building on the wind flow when different façade configurations are considered. In order to assess this influence, two-dimensional (2D) and a three-dimensional (3D) CFD analyses were employed.

The analysis started considering a perfect square as a benchmark shape. This shape was altered by adding setbacks or wedges at the corners of the building to observe significant differences in the average and time-histories of the aerodynamic coefficients. The effect of these façade elements on the wind flow at the pedestrian level was examined through 3D CFD analysis. The comfort of pedestrians was assessed against the Wind Microclimate Guidelines for City of London.

Keywords: Pedestrian Comfort, Building Shape, CFD

1. INTRODUCTION

Extreme wind actions on tall structures consist a major concern when considering the design, the future comfort and serviceability criteria of tall structures, and their effects on the urban environment. The shape, size, the building arrangement and spacing within an urbanised environment, are some of the main parameters that affect the wind flow and structural behaviour. The influence on the dynamic loading can be controlled by two approaches, the “Aerodynamic Mitigation” technique and the “Aerodynamic shape optimisation” technique. The present study focuses on the latter. Merrick and Bitsuamlak (2009) investigated different wind loading patterns for widely used building footprints. Five models with commonly used building shapes were considered for this experimental study in a low-speed boundary layer wind tunnel (BLWT), for which the Reynolds number (Re) was approximately 150,000. The results obtained from this research refer to the general wind behaviour of simplified building shapes, indicating that certain shapes are more prone to wind-related problems like vortex shedding. Rej and Bairagi (2020) present a detailed study of how the sharp edges of tall structures affect the structure and the wind field around them. According to this work cutting out the corners or establishing setbacks methods can reduce the wind-induced vibrations in the building. Furthermore, Elshaer et al. (2017) demonstrated that the limitation of the wind load can be achieved by carefully shaping the corners of the building. The present work continues the lines established in those papers by proposing different façades shapes that can be potentially achieved using moving screens in a future study.

2. METHODOLOGY

Three different building footprints were examined and compared in regards to their aerodynamic coefficients which were recorded via a 2D CFD analysis. Through this, the effect of the shape of the building in the transient flow conditions around it was examined. Further to this the pedestrian's comfort was studied by means of 3D CFD analysis.

The case of a single plate at the edges of a building was analysed and compared with different shapes shown in Figure 2. Series of two and three-dimensional (2D & 3D) CFD finite volume analyses were conducted in OpenFOAM-v2106 (2021). In the CFD analysis the Reynolds-averaged Navier–Stokes (RANS) equations and the standard k–ε turbulence model were solved in transient in the case of 2D, and the steady state solver is used in the 3D CFD for computational efficiency. The pressure-velocity coupling of the fluid motion equations were solved with the semi-implicit algorithm SIMPLE in the 3D CFD, whilst the PISO pressure-implicit algorithm is used in 2D. For the 2D, the boundary conditions chosen set the outlet as pressure outlet, whilst the top and bottom were assigned as symmetric planes. In the case of the 3D CFD the top, front and back patches were set to be empty, while the bottom was set to be the ground (no-slip wall). In both of the 2D and 3D analyses the building obstacles were assigned to meet non-slip wall conditions.

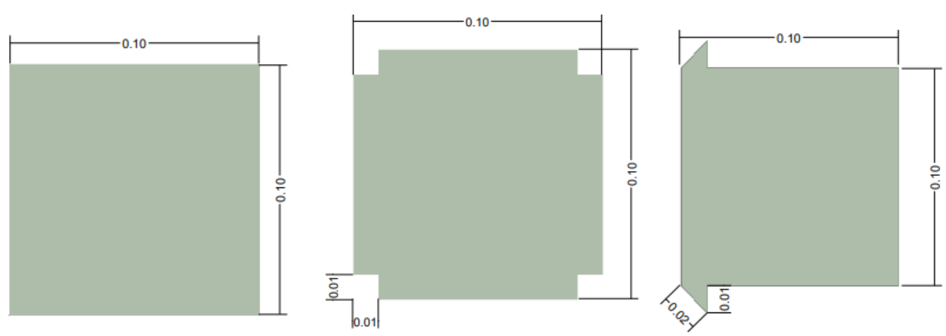

Figure 2. CFD Geometries; a) Simple Square, b) Square-setbacks, c) Square-plates, (dimensions in metres)

The mesh, with near wall layer thickness equal to 5.e-005 m for the 2D and 1e-3 m for the 3D, was carefully refined around the obstacles as well as within the Von Karman street as shown in figures 3 and 4. For the 2D case the ANSYS Fluent (2018) meshing tool was used and for the 3D cases the snappyHexMesh utility of the OpenFoam was chosen. For the 2D cases the reference length, the along the wind flow obstacle's geometrical dimension, was standardised and hence the relevant wall functions for k and ε were 0.375 and 0.954 respectively, while the inlet wind speed was considered to be 10 m/s. All 2D and 3D shapes considered for this study have a Reynolds number equal to Re=66,667 and Re= 70,267 respectively. Similar approach was adopted for the 3D cases-by also including the atmospheric boundary layer criteria as defined by the EN 1991-1-(2010) Section 4 and its UK National Annex. The wall functions k and ε were 0.071 and 0.935

respectively. The terrain was chosen to be category IV which corresponds to an urban environment, the non-scaled reference wind speed was set to be 10 m/s at a height of 10m, as suggested by the EN 1991-1-(2010) Section 4. The reference wind speed was considered to be at the top of the scaled 3D building, H=0.4m, considering the scale used to represent the physical aspect ratio (H/B)-equal to 4- of a high-rise structure in the 3D CFD analysis, where H-is the height and B the reference width. The actual height of the building is and H_{actual}=92m.

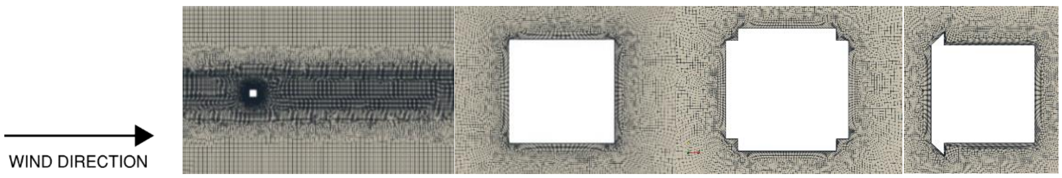

Figure 3. Mesh around the building plan cross-sections in the 2D CFD models

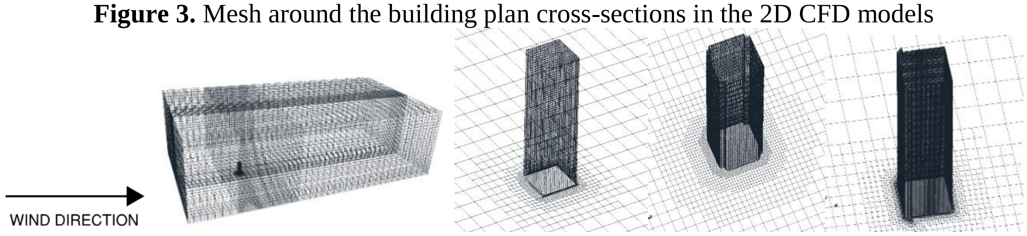

Figure 4. Mesh around the building in the 3D CFD models

3. CFD RESULTS

Comparing the velocity (U_x) displayed in Fig 5. It is observed that the setbacks, and thereafter the plates cases lead to a velocity bubble behind and around the obstacle that decreases in size, when compared to the benchmark.

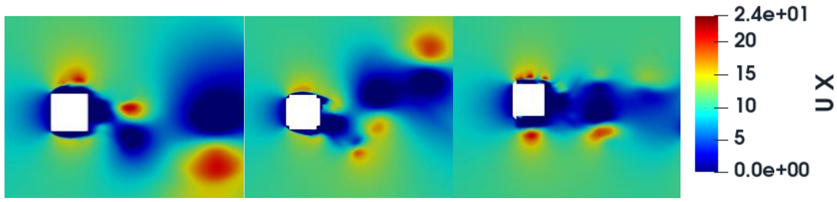

Figure 5. Velocity Contours U_x in m/s from 2D CFD analysis (left to right; Benchmark, Set-backs, Plates)

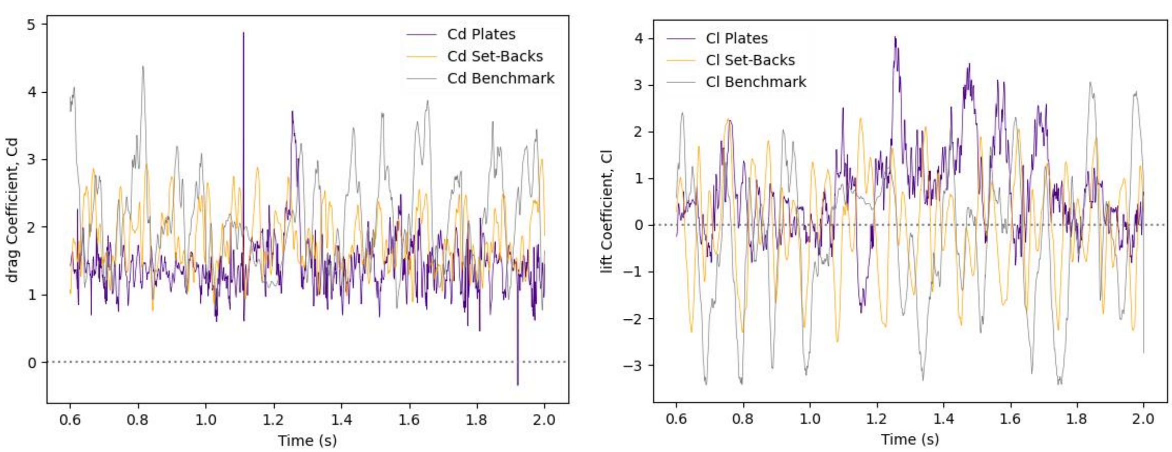

Figure 6. Drag and Lift Coefficient time histories for the examined-via 2D CFD-building shapes.

The drag coefficient produced by the plates shown in Fig 6 is much lower than the benchmark and the setback cases. This suggests that the along-wind vibrations in the case of the plates could be lower and therefore that installing triangular plates at the corners of the building could improve its users' comfort. The same observation applies to the case of the lift coefficient, the plates return lower than the rest of the cases lift coefficient signal. The lift coefficient is associated with the across wind vibrations, which significantly influence the occupant comfort criteria.

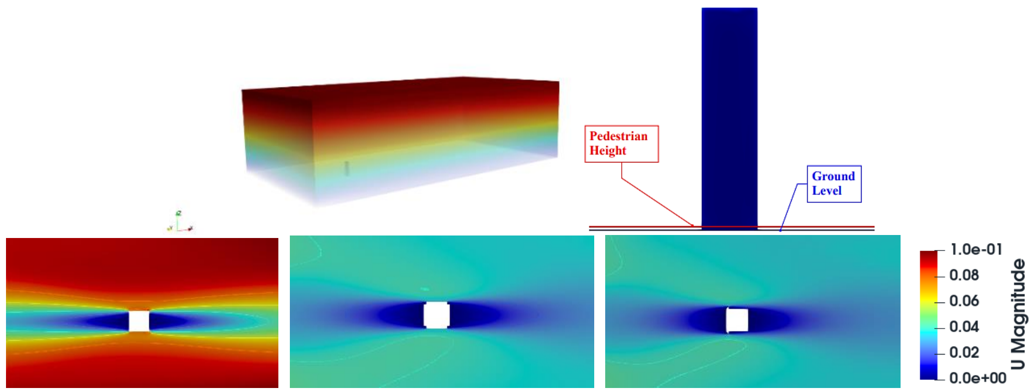

Figure 7. Velocity magnitude contours taken at a height of 0.0065 m (which is the scaled value of the 1.5m height threshold)

Figure 7 presents the 3D CFD velocity magnitude contours around the buildings. The blue coloured velocity values presented in this figure indicate safe wind sped levels, while the red corresponds to extreme ones. In figure 7 the case of the plates reduces the mean wind velocity contour at the examined area when compared with the squared and the setback shapes More specifically the area with wind velocity values in the range from 0 to to 0.02 m/s is 4.5% and 15% larger than that in the model with set-backs or with a perfect square shape, respectively.

Vibrations induced by strong winds cause discomfort to the users of tall buildings as well as vortex shedding that can affect buildings and people in the surroundings. From the above results it is demonstrated that the installation of the proposed façade elements at the corners of the building could improve its users' comfort, whilst ensuring a safer environment to the pedestrians that use the nearby streets.

REFERENCES

ANSYS Fluent (2018). ANSYS Engineering Analysis System Use's Manual. Houston.

City of London Corporation (2019). City of London Wind Microclimate Guidelines. [pdf] Available at: https://www.cityoflondon.gov.uk/assets/Services-Environment/wind-microclimateguidelines. pdf [Accessed 2 December 2021].

Elshaer, A., Bitsuamlak, G. and El Damatty, A., 2017. Enhancing wind performance of tall buildings using corner aerodynamic optimization. Engineering Structures, 136, pp.133-148.

EN 1991-1-4:2005+A1:2010 Section 4 & UK National Annex to BS EN 1991-1-4:2005+A1:2010

Merrick, R. and Bitsuamlak, G., 2009. Shape effects on the wind-induced response of high-rise buildings. Journal of wind and engineering, 6(2), pp.1-18.

OpenFOAM, version 2106; software. Available at https://www.openfoam.com.

Pal, S., Raj, R. and Anbukumar, S., 2021. Comparative study of wind induced mutual interference effects on square and fish-plan shape tall buildings. Sādhanā, 46(2), pp.1-27.

Rej, A. and Bairagi, A.K., 2020. Wind load analysis of a tall Structure with Sharp and Corner Cut Edges. In Advances in Structures, Systems and Materials (pp. 175-183). Springer, Singapore.