20th Australasian Fluid Mechanics Conference
Perth, Australia
5-8 December 2016

# Numerical Study of Flow Pattern and Pedestrian Level Wind Comfort Inside a Uniform Street Canyon at Different Angles of Attack

**P. P. Pancholy**[1], **K. Clemens**[2], **P. Geoghegan**[2], **M. Jermy**[2], **M. Moyers-Gonzalez**[1] **and P. L. Wilson**[1]

[1]School of Mathematics and Statistics
University of Canterbury, Christchurch 4800, New Zealand

[2]Department of Mechanical Engineering
University of Canterbury, Christchurch 4800, New Zealand

## Abstract

The air flow around an individual building is complex. Around two or more buildings, a recirculating flow can occur in the street canyon between them. Such urban flows can introduce high wind speeds at pedestrian level in the building wake or in the recirculating flow in the street canyon, causing discomfort or even injuries. In this paper, a fundamental study using the steady Reynolds-Averaged Navier-Stokes (RANS) approach of computational fluid dynamics (CFD) simulations has been carried out to study the effect of different flow angles of attack (AOA) on the flow pattern and pedestrian comfort inside a uniform street canyon. Analysis was performed at 0°, 15°, 30°, 45°, 60°, 75° and 90° AOA using turbulent conditions. The Reynolds number involved in this study was $8.1 \times 10^6$ based on the height of the building and free stream velocity. A street width to building height aspect ratio of $S/H = 2$ has been considered in this study because high spacing between buildings could be considered dangerous for pedestrian comfort. The aim of this study is to provide input into knowledge-based expert systems by providing mean wind speeds at the entire pedestrian level street width. This study reveals that at a given separated distance of buildings inside a street canyon pedestrian comfort is greater when flow approaches angles of 0°, 15°, 60° and 75° compared to other AOA. It also reveals that the flow structure inside a street canyon at a given aspect ratio is different to the single building case: entrainment of the surrounding fluid towards the axis of symmetry by the horseshoe vortex in the wake region of the buildings or inside the street canyon, loses symmetry.

## Introduction

High rise buildings in urban areas should be designed to ensure comfort of their inhabitants and users. The construction of a building inevitably changes the outdoor environment around the building. These changes include wind speed, wind direction, air pollution, driving rain and heat radiation. The change of these quantities depends on the shape, size and orientation of the building and on the interdependence of the buildings with surrounding buildings [3].

General flow features around a single building when the wind flow approaches a perpendicular angle to the building, include flow separation upstream of the windward face of the building, formation of a stagnation point on the windward face of the building (with maximum pressure at that point), deviation of the flow into four main streams around the building from the stagnation point, separation of the flow due to the sharp windward edge of the building roof and recirculation and reattachment of the flow in the wake of the building. Where the windward wall of the building meets the ground, flow separation forms a horseshoe vortex which escapes around either side of the building and entrains surrounding fluid towards the axis of symmetry in the wake of the building.

When two or more buildings are considered lined up along two sides of a street, they create a *street canyon* in-between, which is vertically bounded by the ground surface and the roof level. The dimensions of a street canyon are expressed by their aspect ratios $S/H$ (street width to building height), and $S/W$ (street width to building width). Note that, in this study upwind and downwind buildings are identical. A street canyon is said to be *uniform* if the adjacent building heights are equal.

Urban areas can be characterized as a group of such street canyons. Wind comfort and wind safety for pedestrians are important requirements for urban areas. This wind comfort and wind safety generally refer to the mechanical effects of wind on people [11]. According to the Beaufort scale of wind effects on people by Lawson et al. [11], at Beaufort Number 3 (gentle breeze or wind speed between $2.4 - 3.8$ m/s) these effects include disturbed hair, clothes flapping and newspaper being difficult to read. So, if we consider a person sitting in an open cafe or standing at a bus stop in the street canyon, wind can cause disturbance. Therefore, in this study a reference wind speed of 5.9 m/s was chosen in such a way that we can analyze wind speed at the pedestrian height of approximately 1.75m.

According to Blocken et al. [4], fundamental studies for pedestrian level wind assessment are typically conducted for simple, generic building configurations to obtain insight into the flow behavior, to study the influence of different building dimensions and street widths, to provide input for knowledge-based expert systems (KBES), and for model validation.

Fundamental studies have been conducted by Ishizaki et al. [10] and Wiren [17], who carried out wind tunnel measurements along the street center line between various two-building configurations. Both studies focused on the mean wind speed in the street between rectangular buildings of equal height. Contours of mean wind speed and turbulence measurements at pedestrian level in streets between two high rise buildings of equal height for parallel and perpendicular wind direction using wind-tunnel experiments were provided by To et al. [14]. Numerical studies for two-building models were conducted by Bottema [6] and Baskaran et al. [2]. A very detailed numerical assessment of the influence of varying a wide range of street widths was first conducted by Blocken et al. [4] for parallel wind direction and with buildings of equal height.

These studies on wind speed conditions in a street canyon were mainly focused on pedestrian-level winds for discrete points, a limited range of street widths and for wind parallel to the street canyon. Detailed CFD study of wind blowing at different AOA to the street still requires more attention. In this study, information has been provided for the mean wind speed at the entire pedestrian level street width for the considered wind directions in order to assess pedestrian wind comfort inside street canyons.

#### CFD simulations: computational model and parameters

Model Description

The model geometry of the uniform street canyon with dimensions $W \times H \times L = 80\,(m) \times 20\,(m) \times 20\,(m)$ was chosen to represent common medium-rise building structures; the chosen street width was $S = 40$ m as shown in Figure 1, which also shows the AOA, $\theta$. The size of the computational domain was selected according to CFD best practice guidelines by Franke [8].

The effect of the changing wind AOA on the flow pattern and pedestrian wind comfort inside a street canyon was investigated by performing CFD simulations. AOA considered in this study was 0°, 15°, 30°, 45°, 60°, 75° and 90°. The Reynolds number was $8.1 \times 10^6$ based on building height (height of the upwind building = height of the downwind building = building height) and free stream velocity.

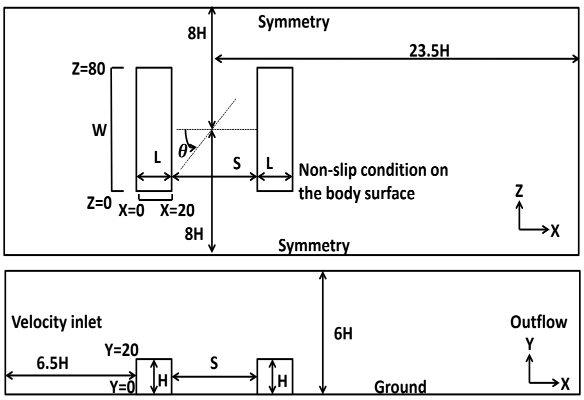



Figure 1: Computational domain and boundary conditions for the street canyon.

Boundary Conditions

Simulations were performed using the commercial CFD package Ansys Fluent version 17.0. The inlet boundary condition was specified according to COST (European Cooperation in the field of Scientific and Technical Research) Action 732 by Franke [8] recommendations and using a user defined function (UDF) satisfying Equations 1– 4 below for the velocity $U(z)$, turbulent kinetic energy $k(z)$ and turbulent dissipation rate $\omega(z)$ respectively:

$$U(z) = \frac{U^*_{ABL}}{\kappa} \ln\left(\frac{z+z_0}{z_0}\right); \quad (1)$$

$$k(z) = \frac{U^{*2}_{ABL}}{\sqrt{C_\mu}}; \quad (2)$$

$$\varepsilon(z) = \frac{U^{*3}_{ABL}}{\kappa(z+z_0)}; \quad (3)$$

$$\omega = \frac{\varepsilon(z)}{C_\mu k(z)}. \quad (4)$$

Here, $\kappa = 0.4$ is the von K*á*rm*á*n constant, $z_0$ is the aerodynamic roughness length, $C_\mu$ is the turbulence model constant and $U^*_{ABL}$ is the atmospheric boundary layer friction velocity, which can be calculated by a specified velocity $U_{ref}$ at reference height $z_{ref}$ as,

$$U^*_{ABL} = \frac{\kappa U_{ref}}{\ln\left(\frac{z+z_0}{z_0}\right)}.$$

Here we take, $U_{ref} = 5.9$ m/s, the free stream wind speed at the building height $z_{ref} = 20$ m to analyze the wind speed between $2.4 - 3.8$ m/s at the pedestrian height.

The top and side boundary conditions were specified as symmetry while the outlet boundary condition was specified as outflow. The bottom boundary condition was specified as a wall. Viscous boundary layers were generated on the ground and building faces with 40 grid layers. The height of the first cell of the boundary layer was chosen to be $1.8 \times 10^{-4}$ m to ensure wall unit $y^+ < 5$ for modeling. The mesh used in this study contains tetrahedral and wedge shaped elements.

Validation

Vardoulakis et al. [16] and Ratnam et al. [13] reported that the most widely studied flow problem in wind engineering is a 3D cube immersed in a turbulent boundary layer due to the simplicity of the shape and the complexity of the flow around the cube. Therefore, we validate our CFD model by first simulating the wind flow around a surface mounted cube in a turbulent channel flow. In this study, comparison has been done for the pressure coefficients $C_p$ along the vertical centerline of the windward face, the roof and the leeward face with the wind tunnel and CFD study results of Irtaza et al. [9] for the Silsoe cube. Our results showed good agreement of approximately 90% in $C_p$ compared to the LES study results of [9].

Computational Mesh

A mesh independence study was carried out to demonstrate the independence of the flow field on the refinement of the mesh for the flow past a cube with dimension $0.2m^3$. The Reynolds number involved in this study was $0.66 \times 10^5$ based on the cube height and free stream velocity. The coarse mesh had 1 million cells of resolution 0.01 m on the faces of the cube and 0.02 m throughout the rest of the computational domain. The medium mesh had 2 million cells with resolution of 0.008 m on the faces of the cube and 0.016 m throughout the rest of the computational domain. The fine mesh had 4 million cells and a resolution of 0.006 m on the faces of the cube and 0.012 m elsewhere. The pressure coefficients $C_p$ were measured along the mid-width of the cube along the upwind face, the top and the downwind face of the cube. The main flow features for the coarse mesh are the same as for the fine mesh. Thus, it can be concluded that the coarse mesh is sufficient for running a mesh independent solution. Therefore, in this study the mesh contains resolution of 1 m on the faces of the buildings and 2 m throughout the rest of the computational domain. The result is a mesh with 6 million cells.

Other Parameters

The transition $k - kl - omega$ (3-equation) model was used for this study. The Pressure-Implicit with Splitting of Operators (PISO) algorithm scheme was used for the pressure-velocity coupling and pressure interpolation was second-order. Second-order discretization schemes were used for both convective terms and viscous terms of the governing equations. The solution was initialized by the values of the inlet boundary conditions. Surface monitor points inside the street canyon with $(X,Y,Z)$ coordinates $(25,12,25)$, $(32,5,40)$, $(40,2,60)$, $(48,14,10)$, $(52,6,52)$ and $(57,4,35)$ were used to measure convergence. These are the points used for AOA 0° and are changed accordingly with change in angle. The simulations were terminated when all specified surface monitor points reached the criteria of a difference in value between two iterations of 0.0005 for 20 consecutive iterations.

Results and Discussion

Figure 2(a) shows the flow pattern at pedestrian height inside the street canyon with the flow approaching at 0° to the windward face of the upwind building. The overall flow pattern observed in this case is similar to the single building case as described previously. However, in this case, the horseshoe vortex, which generally entrains surrounding fluid towards the axis of symmetry in the wake region of the building or inside the street canyon, loses symmetry. According to Martinuzzi et al. [12], when the obstacle separation is between 1.5H to 2.5H, periodic vortex shedding inside the street canyon can be triggered by the interference between a vertical flow stream along the front face of the downstream obstacle and the vortex in the canyon. They observed this phenomena in the case of a laminar boundary layer of thickness approximately 0.07H. In this work the case is a turbulent boundary layer flow with boundary layer thickness of 0.016H. However, according to Castro et al. [7], if the boundary layer thickness is less than approximately 70% of the body height, vortex shedding may be observed. Therefore, it is justified that vortex shedding can be observed in the case of flow approaching at 0° to the street canyon. The reason why steady RANS CFD is not capable of reproducing the vortex shedding in the wake of buildings or inside street canyons is due to the underestimation of turbulent kinetic energy in these regions [5]. Therefore, it is desirable to use unsteady RANS or Large Eddy Simulations (LES) for highly accurate CFD analysis. However, in order to use these models for predicting wind environment around buildings, a dramatic increase in computer processing speed is needed. Unsteady RANS modeling was used for validation study for the single cube case and a total of 16CPUs were used in parallel for the simulation. It takes approximately 10 days to get desired results.

Flow patterns inside the street canyon vary drastically at the AOA 30° and 60° (Figure 2(b-c)). In the case of 90° (Figure 2(d)), flow patterns around both buildings are somewhat independent, and behave like that of a single building.

Figure 3 shows contour plots of pedestrian comfort inside the street canyon at different AOA. For 0°, most of the area around the buildings, which would be in frequent use by pedestrians is in the comfort zone; at AOA 15° this area reduces and for 30° and 45° approximately 95% of the street area is in the uncomfortable zone due to the increased wind speed. Improvement near the leeward wall of the upwind building can be seen in the case of 60° and 75° with a greater area in the comfort zone, but this again disappears at 90°. So, for this configuration, comfort of pedestrians may require wind barriers at pedestrian level or changing the roof shape of the buildings.

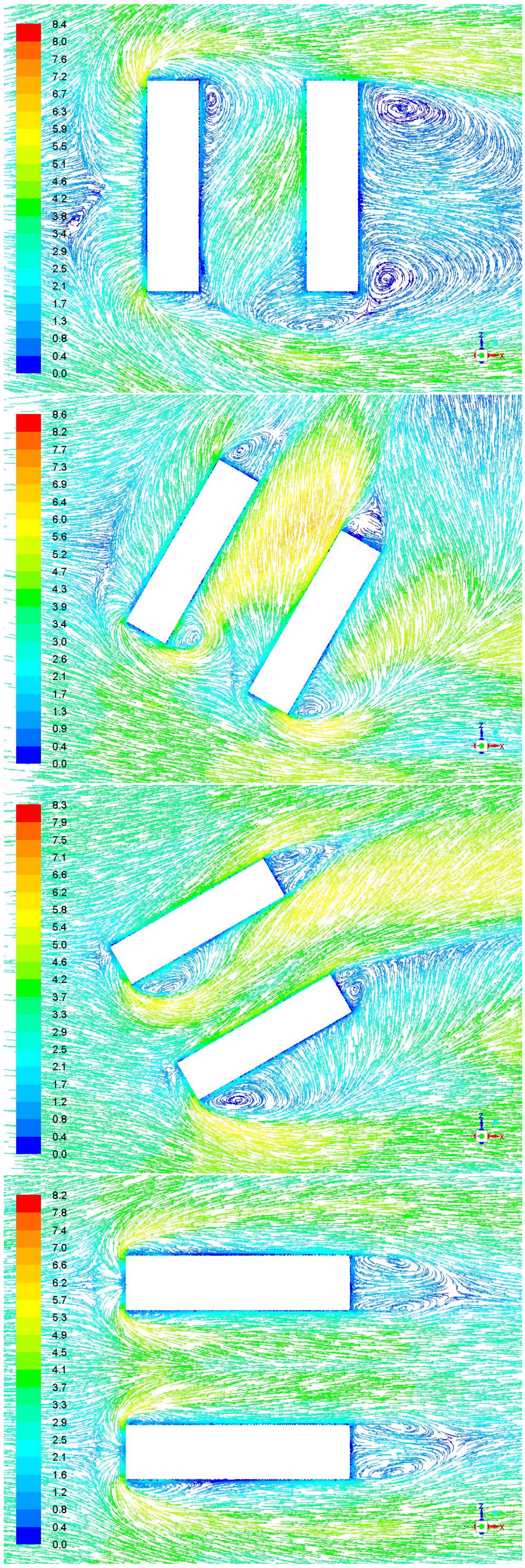
Wind flow pattern pathlines and pedestrian comfort contour plots at 15°, 45° and 75° will be discussed in more detail in future work.

### Conclusions

The results presented in this paper for the mean speed for the entire pedestrian level inside the street canyon with $S/H = 2$, clearly shows that pedestrian comfort is most prevalent in the case of an AOA of 0°, and as the AOA approaches 90°, wind speed increases and regions of discomfort are found inside the street canyon.

### Acknowledgements

The authors wish to acknowledge the contribution of the high-performance computing facilities available at the University of Canterbury to the results of this research and funding from IBM grant number E6327.



(a)

(b)

(c)

(d)

Figure 2: Velocity magnitude (m/s) pathlines in $XZ$- plane at pedestrian level height (1.75m) for angle of attack (a) 0° (b) 30° (c) 60° (d) 90°.

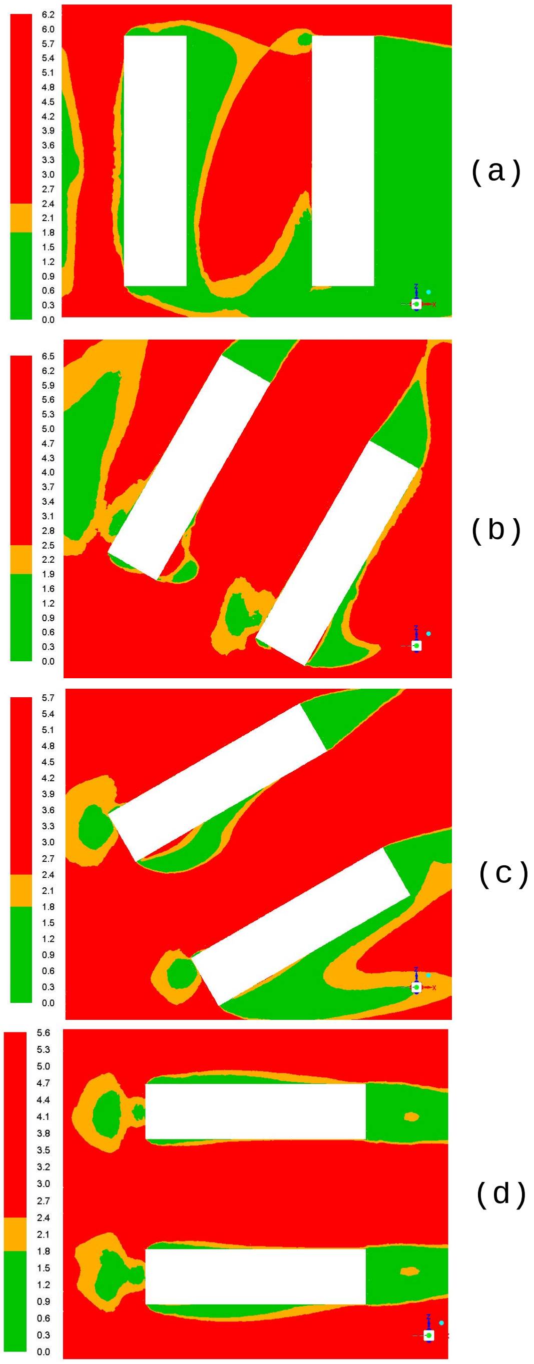



Figure 3: Velocity magnitude (m/s) contours in $XZ$- plane at pedestrian level height (1.75m) for angle of attack (a) 0° (b) 30° (c) 60° (d) 90°, green represents 0 to 1.8 m/s (which can be considered a comfort zone for pedestrians), orange represents 1.81 to 2.4 m/s (zone in which the wind starts causing impact on people). Red represents wind speeds above 2.4 m/s.

### References

[1] Abohela, Islam, Neveen Hamza, and Steven Dudek.“Effect of roof shape, wind direction, building height and urban configuration on the energy yield and positioning of roof mounted wind turbines.” Renewable Energy 50 (2013): 1106-1118.

[2] Baskaran, Appupillai, and Ahmed Kashef. “Investigation of air flow around buildings using computational fluid dynamics techniques.” Engineering Structures 18.11 (1996): 861-875.

[3] Blocken, Bert, and Jan Carmeliet. “Pedestrian wind environment around buildings: Literature review and practical examples.” Journal of Thermal Envelope and Building Science 28.2 (2004): 107-159.

[4] Blocken, Bert, Jan Carmeliet, and Ted Stathopoulos. “CFD evaluation of wind speed conditions in passages between parallel buildingseffect of wall-function roughness modifications for the atmospheric boundary layer flow.” Journal of Wind Engineering and Industrial Aerodynamics 95.9 (2007): 941-962.

[5] Blocken, Bert, et al. “Application of computational fluid dynamics in building performance simulation for the outdoor environment: an overview.” Journal of Building Performance Simulation 4.2 (2011): 157-184.

[6] Bottema, M. Marcel. Wind climate and urban geometry. Diss. Technische Universiteit Eindhoven, 1993.

[7] Castro, I. P., and A. G. Robins. “The flow around a surface-mounted cube in uniform and turbulent streams.” Journal of fluid Mechanics 79.02 (1977): 307-335.

[8] Franke, J. (Ed.). (2007). Best practice guideline for the CFD simulation of flows in the urban environment. Meteorological Inst..

[9] Irtaza, H., et al. “Comparison of wind pressure measurements on Silsoe experimental building from full-scale observation, wind-tunnel experiments and various CFD techniques.” International Journal of Engineering, Science and Technology 5.1 (2013): 28-41.

[10] Ishizaki, Hatsiio, and In Whan Sung. “Influence of adjacent buildings to wind.” Proceedings 3rd International Conference on Wind Effects on Buildings and Structures. 1971.

[11] Lawson, T. V., and A. D. Penwarden. “The effects of wind on people in the vicinity of buildings.” Proceedings 4th International Conference on Wind Effects on Buildings and Structures, Cambridge University Press, Heathrow. 1975.

[12] Martinuzzi, Robert J., and Brian Havel. “Vortex shedding from two surface-mounted cubes in tandem.” International journal of heat and fluid flow 25.3 (2004): 364-372.

[13] Ratnam, G. Seeta, and S. Vengadesan. “Performance of two equation turbulence models for prediction of flow and heat transfer over a wall mounted cube.” International Journal of Heat and Mass Transfer 51.11 (2008): 2834-2846.

[14] To, A. P., and K. M. Lam. “Evaluation of pedestrian-level wind environment around a row of tall buildings using a quartile-level wind speed descripter.” Journal of wind engineering and industrial aerodynamics 54 (1995): 527-541.

[15] Tominaga, Yoshihide, et al. “AIJ guidelines for practical applications of CFD to pedestrian wind environment around buildings.” Journal of wind engineering and industrial aerodynamics 96.10 (2008): 1749-1761.

[16] Vardoulakis, S., Dimitrova, R., Richards, K., Hamlyn, D., Camilleri, G., Weeks, M., ... and Moussiopoulos, N. (2011). Numerical model inter-comparison for wind flow and turbulence around single-block buildings. Environmental Modeling and Assessment, 16(2), 169-181.

[17] Wiren, B. G. “A wind tunnel study of wind velocities in passages between and through buildings.” Proceedings of the 4th International Conference on Wind Effects on Buildings and Structures,(Heathrow 1975). 1975.