The effect of airflow guiding components on effective ventilation rates in single-sided ventilation applications

Nima Najafi Ziarani[1,3], Malcolm Cook[2], Paul D O'Sullivan[*3]

1 MaREI,- SFI Research Centre for Energy, Climate and Marine research and innovation

Ringaskiddy, Cork, Ireland

2 School of Architecture, Building and Civil Engineering
Loughborough University

Ashby Road, Leicestershire, United Kingdom

3 Munster Technological University

Bishopstown, Cork, Ireland

**Paul.OSullivan@mtu.ie*

ABSTRACT

Wind-driven single-sided ventilation (SSV) is present in many existing buildings across Europe and with new Near Zero Energy Building (NZEB) regulations for the refurbishment of the existing building stock, its attractiveness as a non-invasive, low energy solution is set to continue. As a strategy, however, in addition to its air change rate capacity, the distribution of fresh air is an important evaluating criterion for its performance. Airflow guiding components located in the external opening that enhance the effectiveness of the wind-driven flow in ventilating the occupied zone could improve the quality of indoor environments. To our knowledge, the literature is sparse on the practical implications for ventilation when adopting guiding components such as louvers, an increasingly popular approach. In the present study, the performance of wind dominant single-sided ventilation was simulated using RNG $k-\varepsilon$ and RSM CFD models, with and without louvers at three building orientations, e.g. windward, parallel, and leeward. The purpose of this study was to investigate whether louvers installed in the opening would improve both effective ventilation rate and the penetration depth of the flow into the indoor space. The performance of SSV was evaluated using the age of air and interpreting the secondary air circulation inside the room affected by louvers. As the result of these investigations a newly configured airflow guiding component was designed and compared to the other cases. Results show louvers can play a crucial role in controlling the secondary air circulation inside the room and they could either improve or worsen the performance of SSV in terms of air-exchange efficiency. It was shown that in most cases if louvers were the cause of incremental changes in turbulent intensity within the indoor space, then they are effective as an air-exchange efficiency improvement strategy.

KEYWORDS

Single-sided natural ventilation, Computational Fluid Dynamics, Airflow Guiding Components, Near-façade Flow

1 INTRODUCTION

In practice, buildings are vulnerable in terms of reliably providing wellbeing and thermal comfort for the occupants. They are vulnerable to external disturbances e.g. heat weaves, power outages etc. In a resilient cooling strategy, the building should be able to respond to system failures and recover to its previous equilibrium condition (Attia *et al.*, 2021). Therefore, given a building which is fully reliant on mechanical ventilation cannot overcome extreme disturbances such as power outage, natural ventilation can bring more robustness to the design which leads to a more resilient cooling system. Natural ventilation is a low energy strategy, whereas mechanical ventilation can constitute 25% of the total energy in such buildings. The disadvantage of natural ventilation is the lack of air to air heat recovery in winter (Heiselberg *et al.*, 2002; Sacht *et al.*, 2016; Schulze *et al.*, 2018).

Natural ventilation can be classified as either Single-Sided Ventilation (SSV) or Cross Ventilation (CV). Although, cross ventilation is generally a robust ventilation concept which can provide higher ventilation rates and more adaptive comfort control, the performance of SSV is more sensitive to variability in system conditions, particularly the geometry, compared to that of cross ventilation (Goethals *et al.*, 2012; Omrani *et al.*, 2017; Wang and Chen, 2015; Wei *et al.*, 2010a). Nevertheless, cross ventilation can only be used in narrow, open-plan buildings (Wang and Chen, 2015). SSV is perhaps the most common form for modern apartment buildings and offices because it has little restriction and can be easily implemented in buildings, thus it is important to understand the natural ventilation performance of such apartments (Mohamed *et al.*, 2011; Wang and Chen, 2015; Wei *et al.*, 2010b). Therefore, evaluating the performance of SSV is of major value as it could be the most vulnerable strategy in natural ventilation and it is common in most buildings, both dwellings and non-residential; and it is necessary to study how the performance of this system can be improved using either passive or active techniques (Kato *et al.*, 2006).

Various AGCs e.g., guide vane, louver, overhang, etc. are able to reduce cooling demand and to make the airflow more appropriate inside the room. It is indicated that a significant improvement in the indoor thermal comfort condition can be achieved by actuating the window intelligently during the summer indoor conditions (Pokhrel *et al.*, 2019). Field measurements reported the slot louver Air Change Rate (ACR) were 6.5% higher compared with the plain opening ACR; and the slot louver ventilation system has led to steadier ventilation rates (O'Sullivan and Kolokotroni, 2017). Shading louvers lead to an increase in ΔC_p for parallel approach flow and a decrease in ΔC_p for perpendicular approach flow (Zheng *et al.*, 2019). Using venetian blinds can cause even a little bit higher airflow rate when it is open, in comparison with no shading case (Argiriou *et al.*, 2002).

In one aspect, louver application can be interpreted as guide vane, which is guiding the outside airflow into the room in more preferable manner. In a wind-tunnel study it was found that for parallel flow locally at the opening, applying guide vane can improve the normalized airflow rate up to 12 times compared to plain opening (Kato *et al.*, 2006). Overhang drastically enhanced the ventilation rate in the windward direction regardless of the wind speed; however, ventilation rate slightly decreases for the leeward and side cases (Park *et al.*, 2016).

Designing a naturally ventilated building often presents greater challenges than a corresponding mechanically ventilated building (Caciolo *et al.*, 2011; Larsen and Heiselberg, 2008). Accurately modelling the windows of buildings is important to quantify airflow in single-sided natural ventilation (Wang *et al.*, 2017). Therefore, assessment of the complex flow field present in SSV is necessary to realise the flow characteristics in this type of natural ventilation. In the present study Computational Fluid Dynamics (CFD) is adopted to predict the flow characteristics in the case of wind-dominant SSV. Numerical modelling provides spatial information, e.g. predictions of fresh air distribution features of SSV flows (el Telbany *et al.*, 1985). It was reported that adopting the Renormalisation Group (RNG) turbulence model, the discrepancy in the determination of the ventilation rate is acceptable and the flow distribution inside the building is accurately predicted (Evola and Popov, 2006).

With this in mind the purpose of this study was threefold:

I. Numerically evaluate the wind driven near-façade flow in a generic isolated surface-mounted cubic room with a single opening at one side of the building façade.
II. Numerically investigate the performance of SSV at three building orientations with and without louvers at the opening.
III. Appraise a novel airflow guiding component at the opening based on near-façade flow features and their interaction with louvers.

As it is mentioned, in order to improve the performance of a system, detailed evaluation of its characteristics is necessary. In natural ventilation, the main characteristics of the system will be determined by the interaction of the external flow and internal flow. Presence of an isolated surface mounted cube inside an atmospheric boundary layer will cause phenomena such as vortex shedding; which is transient in essence and the present study uses steady-state simulation, therefore the main focus is on the mean values of the near-facade flow characteristics and transient features of the flow are out of context of this study albeit they are of major importance.

2 MATERIALS AND METHODS

CFD simulations were conducted using ANSYS FLUENT 2021R1 which is a finite volume, general purpose code. Two types of turbulence model were used: Reynolds Averaged Navier Stokes (RANS), and Reynolds Stress Model.

The present study used the previous atmospheric boundary layer wind tunnel measurements by Kosutova et al (Kosutova *et al.*, 2019) for CV in a generic surface mounted scaled cubic building (H=150 mm × H=150 mm × H=150 mm) with openings (h=40 mm × w=70 mm) at the centre of façades for validation of adopted turbulent methods. The thickness of the building was 10mm and three louvers with thickness of 0.75mm were applied at the openings with 15 degrees inclination to the horizontal. Streamwise mean velocities were measured at four vertical lines inside the room (Figure 1-d) using particle image velocimetry (PIV).

The inlet velocity profile in the wind-tunnel was reported as following a logarithmic equation:

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

where z is the height in the z-direction (vertical direction from the ground), $u^*_{ABL} = 0.195\ m/s$ is the friction velocity, $\kappa = 0.42$ is the Von Karman constant, and $z_0 = 0.0024$ is the roughness parameter (Kosutova *et al.*, 2019). The measured streamwise turbulent intensity, ($I_u = \frac{\sigma_u}{U_{ref}}$), was 10%.

2.1 Computational domain

The computational domain size for the lateral sides (5H), downstream side (15H), and top boundary (5H) was adopted based on best practice guidelines (Franke *et al.*, 2007) as illustrated in Figure 1-a and 1-b. The upstream boundary was selected as 3H to accommodate changes in inlet velocity profile based on the recommendation of previous studies in RANS modelling (Blocken, 2015). In the present study the building hieght was considered as the reference height, H, and the velocity at building height was considered as reference velocity (U_{ref}).

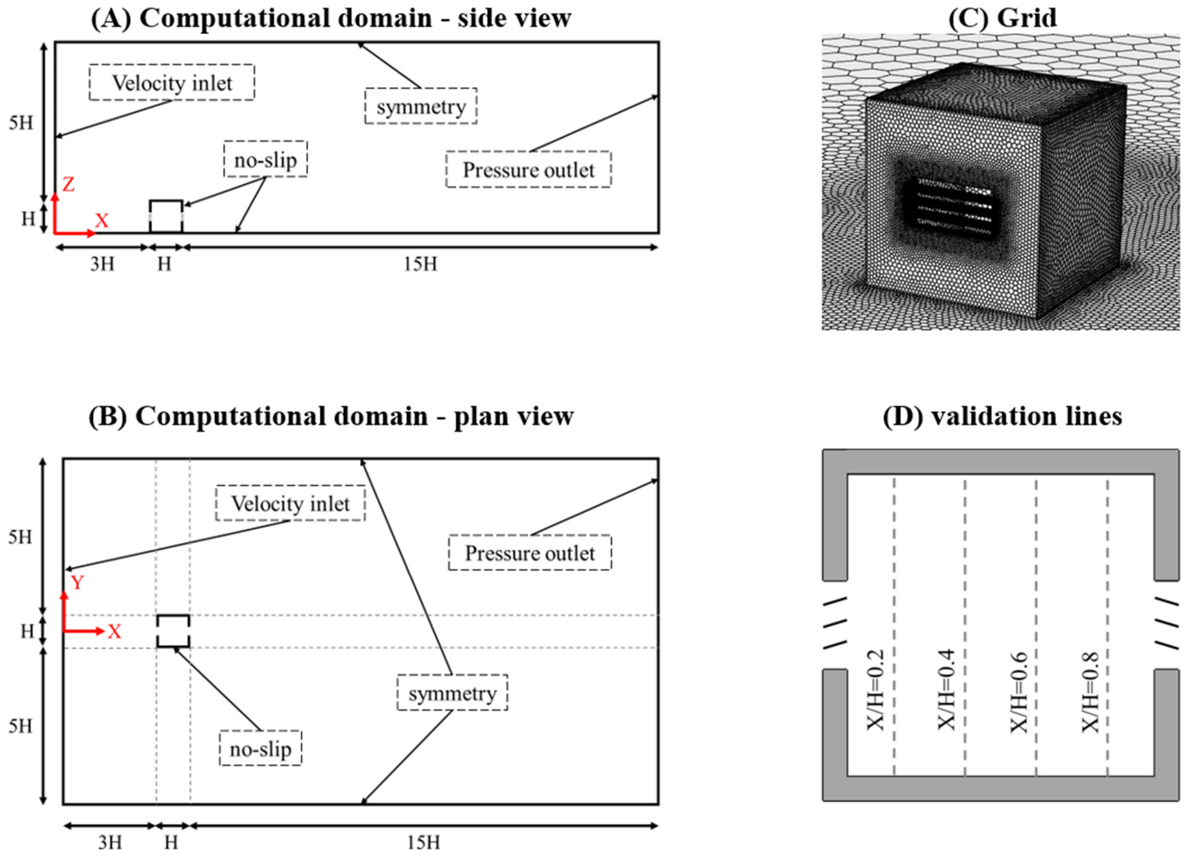


Figure 1- a, b) Computational domain and boundary conditions in the present study, c) poly-hederal grid applied in the present study, d) validation lines provided by previous wind-tunnel study (Kosutova *et al.*, 2019)

2.2 Turbulence modelling

In the present study it is assumed that the flow is steady state, incompressible, three-dimensional and isothermal (as it is wind-dominant natural ventilation, the effect of buoyancy was ignored). Therefore, the continuity equation for mass transfer, the Navier-Stokes equation for momentum transfer are solved to simulate the flow field. Turbulence was modelled using both a Reynolds Stress Model (RSM) (Pope, 2001), and Renormalized-Group (RNG) $k-\epsilon$ method (Yakhot *et al.*, 1992). Since the y^{+}value for building and louvers in all cases was set to be ≈ 1, and this value is higher on the ground surfaces, due to reduction of computation cost, an enhanced wall function was adopted for both RNG $k-\epsilon$ and RSM models to enable flexibility on near-wall grid resolution.

2.3 Mean age of air, air exchange efficiency, and airflow rate calculations

The mean age of air was obtained by solving a transport equation added using a user-defined function (UDF) along with both momentum and continuity equations. The air exchange efficiency, ϵ_A, for all cases was then calculated based on Eq. (2) (Sandberg, 1992):

$$\epsilon_A = 100.\frac{\tau_n}{2<\tau>} \qquad (2)$$

where τ_n is nominal time constant, and $<\tau>$ is mean age of all air present in the room. The simplest method for calculating the airflow rate adopted by many numerical studies, (Caciolo *et al.*, 2012; Jiang *et al.*, 2003), which does not consider turbulent diffusion at the opening, is by integrating the time-averaged normal velocities over the opening area:

$$Q_{vent} = Q_{in} = Q_{out} = \frac{1}{2} \sum_{c}^{N_{cells,opening}} |U_n|.A_c \tag{3}$$

where the U_n is the velocity vector normal to the opening, and A_c is the area of the cell c of the opening.

2.4 Boundary conditions and discretization

The logarithmic velocity profile (Eq. (1)) was applied at the inlet boundary (Figure 1-a and 1-b). In order to apply the turbulent features of the inlet profile to represent 10% measured turbulent intensity, the value of turbulence kinetic energy k was obtained as per Eq. (4), assuming $\sigma_u^2 \approx \sigma_v^2 + \sigma_w^2$ (Kosutova *et al.*, 2019) and therefore $k = \sigma_u^2$.

$$k = \frac{1}{2}(\sigma_u^2 + \sigma_v^2 + \sigma_w^2) \tag{4}$$

The value of the turbulence dissipation rate was determined based on the following (Richards and Hoxey, 1993):

$$\epsilon = \frac{(u_{ABL}^*)^3}{\kappa(z + z_0)} \tag{5}$$

Zero static gauge pressure was applied at the outlet boundary; no-slip boundary condition for the building's walls, louvers and ground, and a symmetry boundary condition for lateral and top boundaries were adopted as demonstrated in Figure 1. The pressure and velocity distribution coupling are obtained using the SIMPLE algorithm. The second order linear upwind scheme is adopted as the discretisation method for pressure, momentum, turbulent kinetic energy, and turbulent dissipation rate. The convergence criterion of the residuals was set to reduce to less than the threshold of 10^{-5} for all equations. The difference between consecutive values of calculated airflow rate at the opening, and volume averaged age of air were monitored to be less than a threshold of 10^{-6} , when the residuals reached their determined criterion to make sure that the parameters of interest were stable.

2.5 Grid study

The computational grid was generated by ANSYS FLUENT 2021R1 meshing using poly-hederal surface mesh. The building geometry and louvers are modelled according to the wind tunnel sample described above. Three grids with a size of 585,063 (coarse grid), 1,734,404 (basic grid), and 5,039,507 (fine grid) respectively were generated to study the effect of grid resolution on the results using Grid Convergence Index (GCI) (Roache, 1997):

$$GCI_{Basic} = F_s \left| \frac{r^p\left[(U_{Basic} - U_{Fine})/U_{ref}\right]}{1 - r^P} \right| \tag{6}$$

where the value of 1.25 is taken for safety factor, F_s, as recommended, the value of $\sqrt{2}$ for r as grid refinement factor, and the value of 2 for p based on the use of second-order discretization schemes for the simulations. The GCI values of 0.98%, 1.68%, 1.69%, and 2.5% for lines x/H=0.2, x/H=0.4, x/H=0.6, x/H=0.8, respectively (Figure 2) for comparison of basic and fine grids show that the effect of grid refinement from basic grid to fine grid is not significant; hence the present study used the basic grid arrangement for evaluating the case-studies.

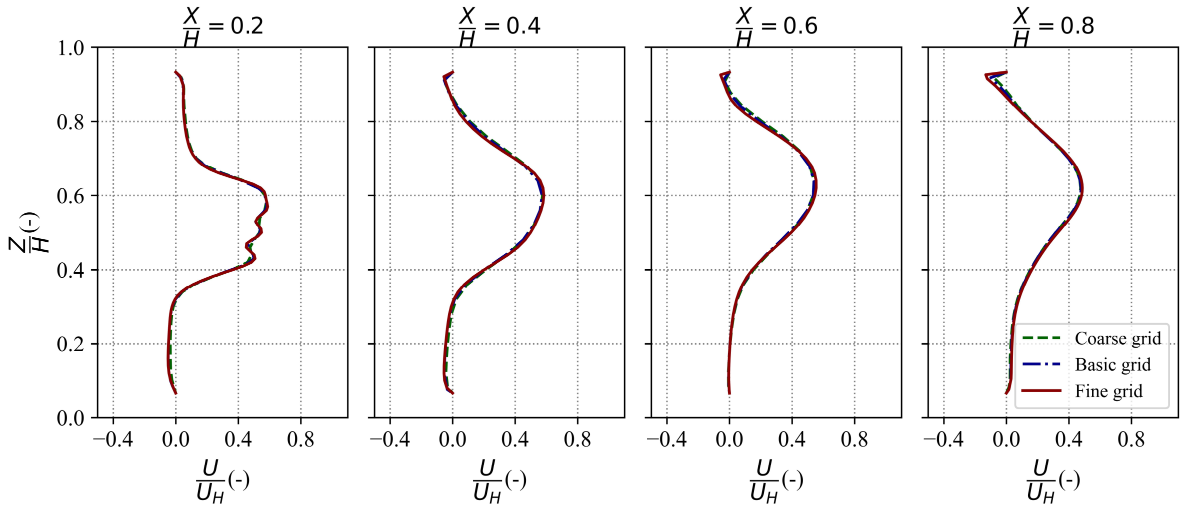


Figure 2- Grid-sensitivity analysis for four vertical lines inside the test room for the cross-ventilation case (x/H lines are illustrated in Figure 1-d)

2.1 Single sided natural ventilation cases considered

To assess the effect of airflow guiding components on the performance of SSV in terms of air exchange efficiency nine separate cases were evaluated; plain opening, louvered opening (as per wind tunnel design(Kosutova *et al.*, 2019)), and newly-designed airflow guiding components (Figure 3-d), when the opening was located at windward (Figure 3-a), parallel (Figure 3-b), and leeward (Figure 3-c) façade.

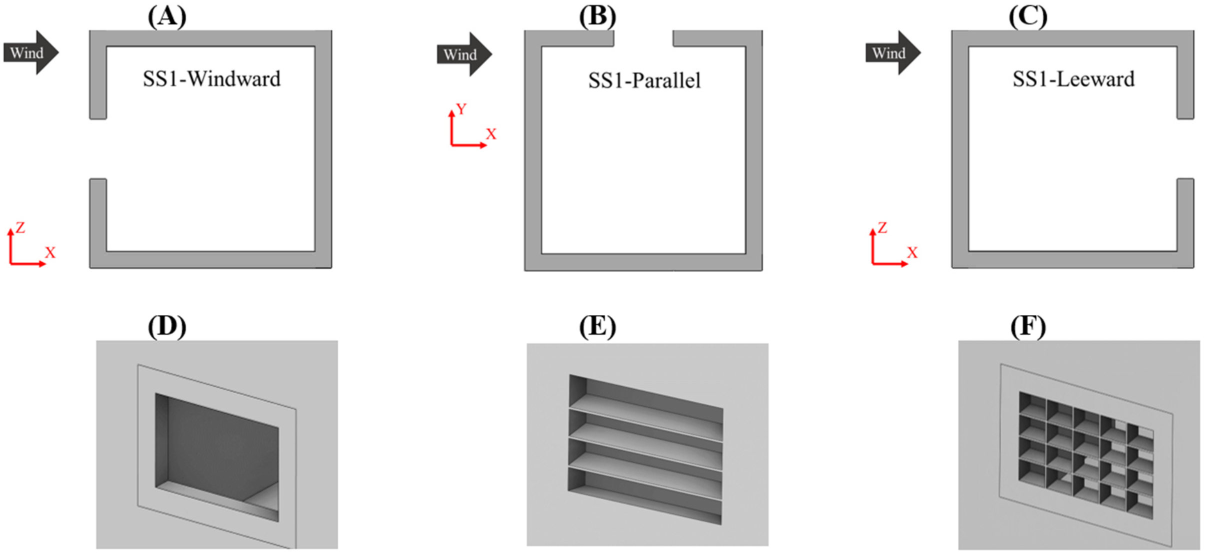


Figure 3- Top: flow directions considered, and, bottom: opening geometries considered

3 RESULTS

3.1 Validation

In order to validate the modelling of the louver geometry at the opening and the accuracy of the adopted turbulent methods, the mean non-dimensional streamwise velocity profiles at four vertical lines as per Figure 1-d obtained using both RNG $k-\epsilon$ and RSM models compared with wind-tunnel results (Kosutova *et al.*, 2019) is illustrated in Figure 4, for cross ventilation. The resulting profiles from both the RNG $k-\epsilon$ and RSM models are in good agreement with the experimental measurements which demonstrates the capability of both models for predicting the airflow distribution in the case of natural ventilation in an isolated surface mounted-cube shaped room with louvers at the opening. Given the more stable convergence behaviour of the RNG $k - \epsilon$ model compared with the RSM for all the cases of SSV, the RNG $k - \epsilon$ model was adopted as the turbulent model in the present study.

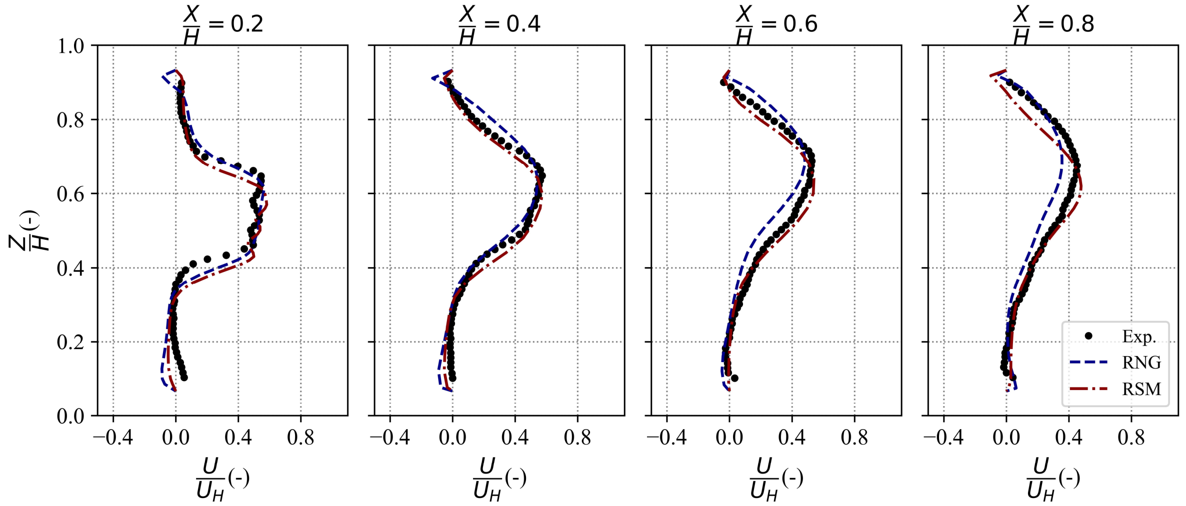


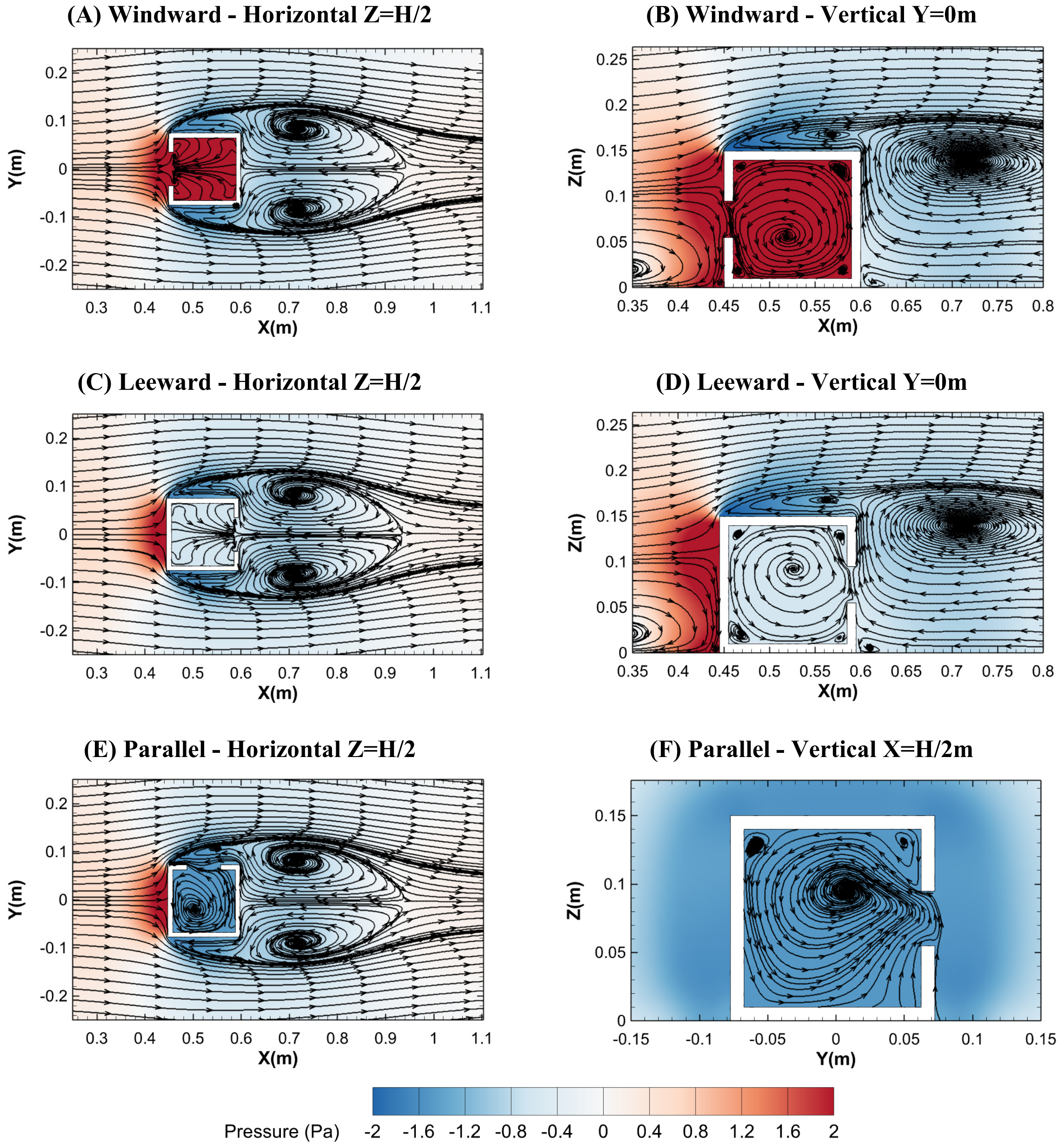
Figure 4- Validation of calculated mean streamwise velocity profiles inside the room compared with wind tunnel study (Kosutova *et al.*, 2019)

3.2 Near-façade flow structures

As shown in Figure 5-a and 5-b on the windward façade there exists an upwash flow (from the stagnation point to the upper edge of the building), a downwash flow (from the stagnation point to the ground), and transversal flows (from the stagnation point towards both side edges of the building). Therefore, even in the windward façade the flow direction is parallel to the opening. This was observed by previous field measurements study (Caciolo *et al.*, 2011; O'Sullivan and Kolokotroni, 2017). Figure 5-c and 5-d, demonstrates the upwash near-façade flow as the result of a low-pressure region on the wake of the building, forming a vortex on the wake side of the building by which the indoor flow is affected from this upwash flow locally at the opening. There are transversal flows also on this side of building façade. On the parallel opening (Figure 5-e and 5-f) however there is a dominant horizontal parallel flow in the opposite direction of the main stream as a result of a low-pressure region caused by the separation on the vertical edges of the building. This leads to a different flow structure inside the room compared to leeward and windward openings. It is noteworthy to mention that the illustrations in Figure 5 are streamlines on the surface, however the flow structures around and inside the room are 3-dimensional and we always have flow in all directions, that is why even on parallel side the streamlines show upwash flow (Figure 6).

3.1 Age of Air and Air-Exchange Efficiency

Figure 7 shows that the mean age of air reduces by the presence of louvers only in the windward side compared to the plain opening. The age of air significantly worsens by the presence of louvers at the opening for the leeward side. The fact that the wind direction is not a constant parameter in real buildings, and an opening can be on the leeward side, windward side or parallel side intermittently due to different wind directions, demonstrates the necessity of designing airflow guiding components in a way that there will be an improvement in all opening orientations in average. As it is discussed in the previous section, the near-façade flow is dominantly parallel locally at the opening.



(A) Windward - Horizontal Z=H/2

(B) Windward - Vertical Y=0m

(C) Leeward - Horizontal Z=H/2

(D) Leeward - Vertical Y=0m

(E) Parallel - Horizontal Z=H/2

(F) Parallel - Vertical X=H/2m

Figure 5- Mean streamlines and mean static pressure contours

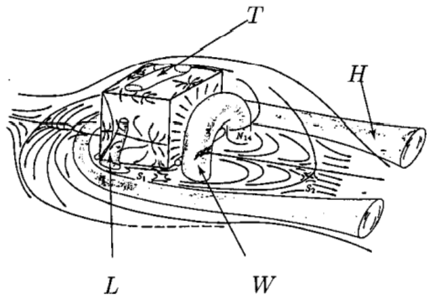


Figure 6- Schematic demonstration of the flow around the surface-mounted cube (Martinuzzi and Tropea, 1993)

On the windward side the downwash flow is dominant, on leeward side the upwash flow, and on parallel side there is horizontal dominant flow. In order to enhance the amount of air entering the room on all three sides, the newly-designed airflow guiding components with both horizontal and vertical fins (Figure 3-f) is applied at the opening to allow an examination of the effect of these components on controlling the near façade flow locally at the opening. As it is clear in Figure 7, the new louver design in terms of age of air has more stable treatment compared to plain and louvered openings. It is noteworthy to mention that the present study does not claim that the suggested newly-designed Airflow Guiding Component (AGC) is the optimum AGC design. It is presented for demonstrating the importance of considering near-façade flow in designing airflow guiding components and effectiveness of this approach on improving the performance of SSV.

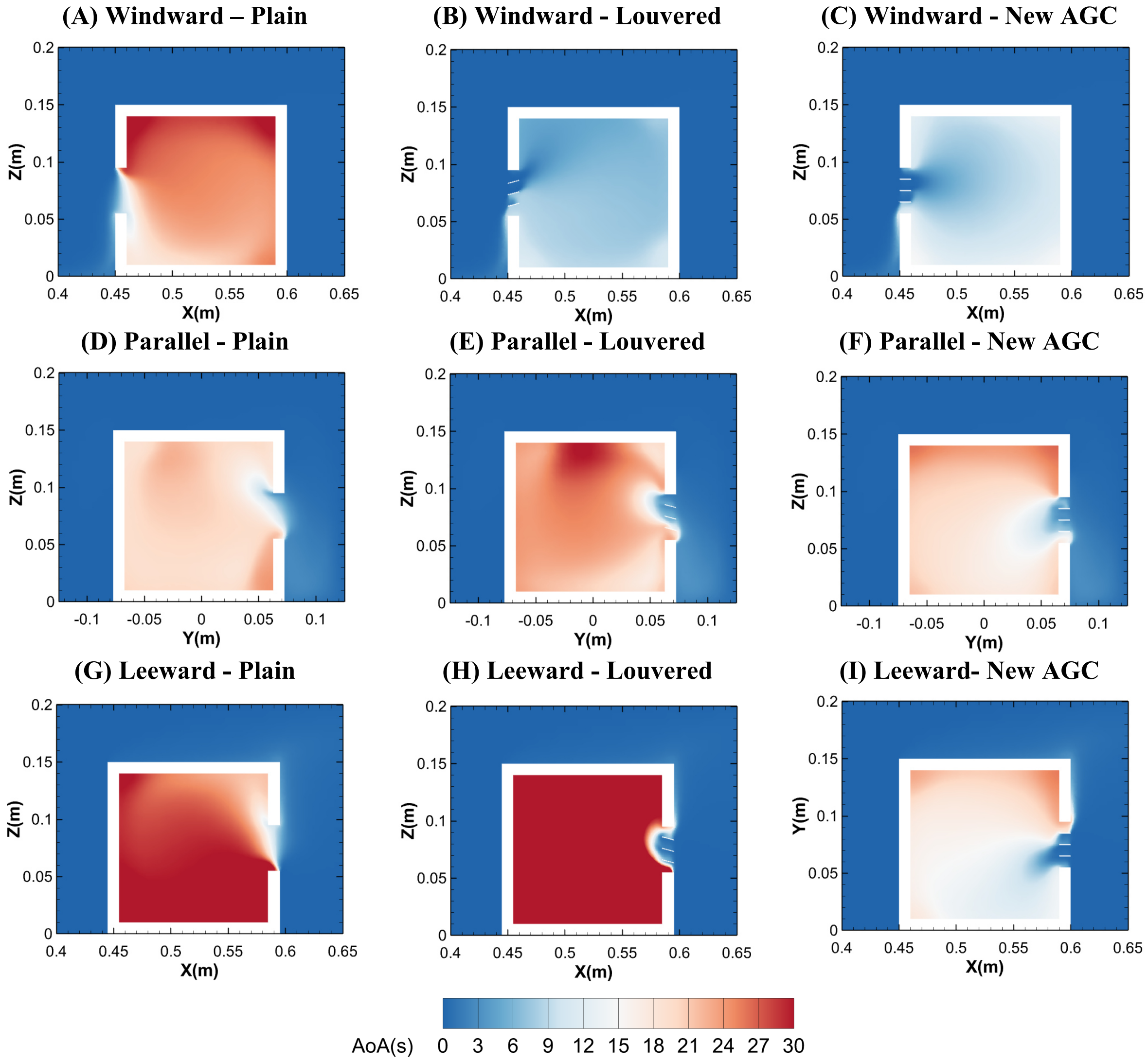


Figure 7- Age of Air contours at vertical surface at the centre of the room

3.1 Effect of louvers and airflow guiding components on indoor airflow

The main question in louver design and the effect they have on the indoor environment is how louvers interact with the near-façade flow and in what way they should be designed to ensure that there will be a positive effect on average for the system's performance. With this approach, the present study looked into the turbulent features of the inlet jet, the relationships between turbulent intensity and air-exchange efficiency, and the jet deflection phenomenon. Figure 8 demonstrates that louvers can cause the jet of air, which enters into the room from the bottom side of the plain opening on the windward side, to be deflected and instead enters from the upper side of the opening. This deflection is the cause of the incremental increase in turbulent intensity of the inlet air and therefore changes in volume-averaged turbulent intensity and air-exchange efficiency (127% improvement), as well as changes of indoor air circulation in this case. The presence of louvers (inclined at an angle of 15 degrees to the horizontal) worsens the air-exchange efficiency on the leeward side compared to the plain opening significantly (50.47% deterioration). The louvers not only cause jet deflection but they are also cause a decrease in volume-average turbulent intensity in this particular case.

Re-evaluating Figure 8 based on the short-circuiting phenomenon illustrates that when louvers are the cause of separating the inlet and outlet air flow paths at the opening, they can be cause of improvement in the performance of the system. Although due to the number of cases the statistically significant relation/correlation between turbulent intensity and AEE cannot be provided in the present study, in most cases when louvers were the cause of increment in turbulent intensity there is an improvement in AEE (Figure 9-d).

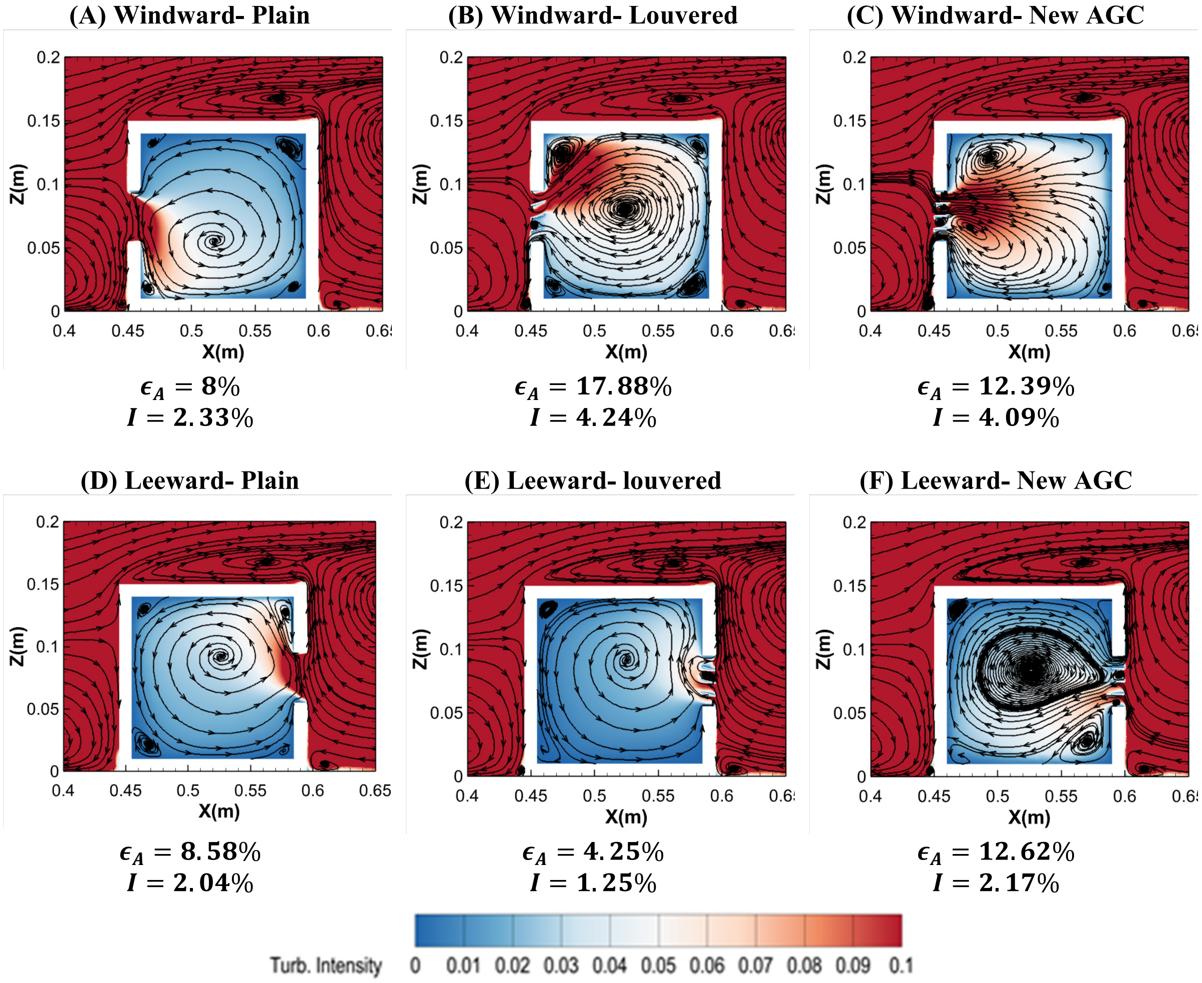


Figure 8- Mean streamlines and turbulent intensity contours on windward and leeward opening

4 CONCLUSIONS

Given the fact that single-sided natural ventilation is one of the worst performing ventilation systems to provide comfort and wellbeing for the occupants, the present study investigated the effect of applying airflow guiding components at the opening in this system to improve the performance of this system as it is present in many existing buildings across Europe. The possibility of providing a new design for these components through an improved understanding of the near-facade flow characteristics was evaluated. It was demonstrated that the structure of the airflow inside the room is determined by the near façade flow in wind dominant single sided natural ventilation and this can be changed and controlled by applying louvers at the opening. The AGCs/louvers not only cause jet deflection but they are also cause a decrease in volume-average turbulent intensity in some particular cases. In most cases when louvers are the cause of increasing turbulent intensity as a result of jet deflection inside the room, there is an improvement in air-exchange efficiency.

Results show that designing AGCs based on the near-façade flow can improve the performance of the SS1 in average. The average air-exchange efficiency in all three orientations (windward, parallel, leeward) is 11%, 11.9%, and 14.43% for plain, louvered, and newly-designed AGCs, respectively. This shows 31.21% improvement on average for the newly-designed AGCs compared to the plain opening, while this figure for louvered opening is only 8.2%.

It was found that when AGCs/louvers are the cause of separating the inlet and outlet air flow paths at the opening, they can be cause of improvement in the performance of the system.

Given the contribution of the newly-designed AGCs to the ventilation performance of the building in wind-dominant single-sided natural ventilation, it is noteworthy to mention that these components can affect the windows view, although, this effect can be seen as a positive effect due to shading and therefore reduction of cooling load during summer, in addition to increasing the security of the building and providing secure possibility of night cooling for the enclosed space.

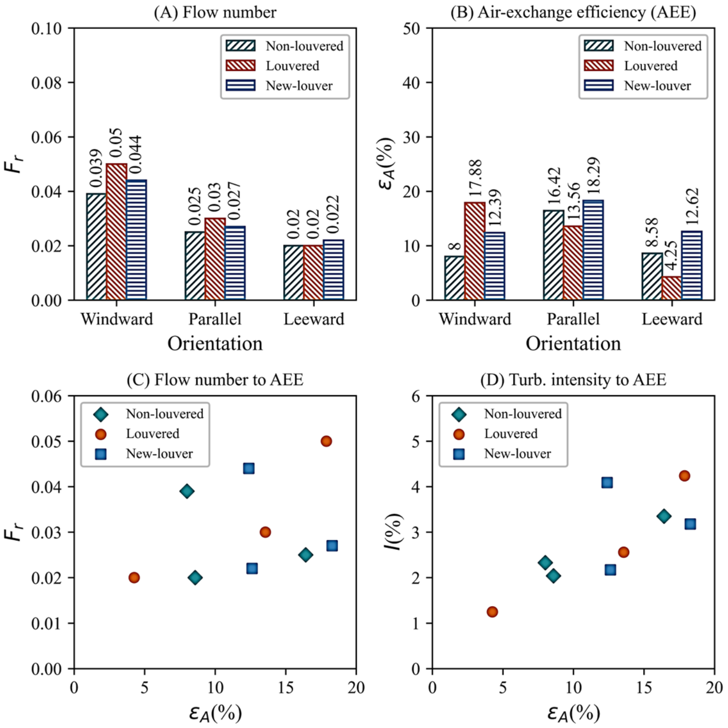


Figure 9- a) Flow number and b) air-exchange efficiency bar charts; c) flow number to air-exchange efficiency, and d) turbulent intensity to air-exchange efficiency scatter plots

5 ACKNOWLEDGEMENTS

This research was part-funded by Science Foundation Ireland (SFI) through MaREI, the SFI Research Centre for Energy, Climate, and Marine, and especially the Centre for Doctoral Training in Energy Resilience and the Built Environment (ERBE) (grant no: 12/R.C./2302_P2, with supporting funding obtained from UK Engineering and Physical Sciences Research Council (EPSRC) (grant EP/S021671/1).

The authors wish to acknowledge the Irish Centre for High-End Computing (ICHEC) for the provision of computational facilities and support.

6 REFERENCES

Argiriou, A.A., Balaras, C.A. and Lykoudis, S.P. (2002), *Single-Sided Ventilation of Buildings through Shaded Large Openings*, *Energy*, Vol. 27.

Attia, S., Levinson, R., Ndongo, E., Holzer, P., Berk Kazanci, O., Homaei, S., Zhang, C., *et al.* (2021), "Resilient cooling of buildings to protect against heat waves and power outages: Key concepts and definition", *Energy and Buildings*, Vol. 239, p. 110869.

Blocken, B. (2015), "Computational Fluid Dynamics for urban physics: Importance, scales, possibilities, limitations and ten tips and tricks towards accurate and reliable simulations", *Building and Environment*, Elsevier, Vol. 91, pp. 219–245.

Caciolo, M., Stabat, P. and Marchio, D. (2011), "Full scale experimental study of single-sided ventilation: Analysis of stack and wind effects", *Energy and Buildings*, Vol. 43 No. 7, pp. 1765–1773.

Caciolo, M., Stabat, P. and Marchio, D. (2012), "Numerical simulation of single-sided ventilation using RANS and LES and comparison with full-scale experiments", *Building and Environment*, Vol. 50, pp. 202–213.

Evola, G. and Popov, V. (2006), "Computational analysis of wind driven natural ventilation in buildings", *Energy and Buildings*, Vol. 38 No. 5, pp. 491–501.

Franke, J., Hellsten, A., Schlünzen, H. and Carissimo, B. (2007), *Best Practice Guideline for the CFD Simulation of Flows in the Urban Environment*, *COST Action*, Vol. 44.

Goethals, K., Couckuyt, I., Dhaene, T. and Janssens, A. (2012), "Sensitivity of night cooling performance to room/system design: Surrogate models based on CFD", *Building and Environment*, Vol. 58, pp. 23–36.

Heiselberg, P., Bjørn, E. and Nielsen, P. v. (2002), "Impact of Open Windows on Room Air Flow and Thermal Comfort", *International Journal of Ventilation*, Informa UK Limited, Vol. 1 No. 2, pp. 91–100.

Jiang, Y., Alexander, D., Jenkins, H., Arthur, R. and Chen, Q. (2003), "Natural ventilation in buildings: Measurement in a wind tunnel and numerical simulation with large-eddy simulation", *Journal of Wind Engineering and Industrial Aerodynamics*, Vol. 91 No. 3, pp. 331–353.

Kato, S., Kono, R., Hasama, T., Ooka, R. and Takahashi, T. (2006), "A wind tunnel experimental analysis of the ventilation characteristics of a room with single-sided opening in uniform flow", *International Journal of Ventilation*, VEETECH Ltd., Vol. 5 No. 1, pp. 171–178.

Kosutova, K., van Hooff, T., Vanderwel, C., Blocken, B. and Hensen, J. (2019), "Cross-ventilation in a generic isolated building equipped with louvers: Wind-tunnel experiments and CFD simulations", *Building and Environment*, Vol. 154 No. October 2018, pp. 263–280.

Larsen, T.S. and Heiselberg, P. (2008), "Single-sided natural ventilation driven by wind pressure and temperature difference", *Energy and Buildings*, Vol. 40 No. 6, pp. 1031–1040.

Martinuzzi, R. and Tropea, C. (1993), "The Flow Around Surface-Mounted, Prismatic Obstacles Placed in a Fully Developed Channel Flow (Data Bank Contribution)", *Journal of Fluids Engineering*, Vol. 115 No. 1, pp. 85–92.

Mohamed, M.F., Behnia, M., King, S. and Prasad, D. (2011), "The potential of natural ventilation in single-sided ventilated apartment to improve indoor thermal comfort and air quality", *ASME 2011 5th International Conference on Energy Sustainability, ES 2011*, pp. 71–78.

Omrani, S., Garcia-Hansen, V., Capra, B.R. and Drogemuller, R. (2017), "On the effect of provision of balconies on natural ventilation and thermal comfort in high-rise residential buildings", *Building and Environment*, Elsevier Ltd, Vol. 123, pp. 504–516.

O'Sullivan, P.D. and Kolokotroni, M. (2017), "A field study of wind dominant single sided ventilation through a narrow slotted architectural louvre system", *Energy and Buildings*, Elsevier Ltd, Vol. 138, pp. 733–747.

Park, J., Choi, J. Il and Rhee, G.H. (2016), "Enhanced single-sided ventilation with overhang in buildings", *Energies*, Vol. 9 No. 3, pp. 1–14.

Pokhrel, M.K., Anderson, T.N. and Lie, T.T. (2019), "Maintaining thermal comfort of a single-sided naturally ventilated model house by intelligently actuating windows", The Association for Computer-Aided Architectural Design Research in Asia (CAADRIA).

Pope, S.B. (2001), "Turublent Flows".

Richards, P.J. and Hoxey, R.P. (1993), "Appropriate boundary conditions for computational wind engineering models using the k-ϵ turbulence model", *Journal of Wind Engineering and Industrial Aerodynamics*, Vol. 46–47 No. C, pp. 145–153.

Roache, P.J. (1997), "QUANTIFICATION OF UNCERTAINTY IN COMPUTATIONAL FLUID DYNAMICS", *Annual Review of Fluid Mechanics*, Vol. 29 No. 1, pp. 123–160.

Sacht, H., Bragança, L., Almeida, M. and Caram, R. (2016), "Study of Natural Ventilation in wind Tunnels and Influence of the Position of Ventilation Modules and Types of Grids on a Modular Façade System", *Energy Procedia*, Vol. 96, Elsevier Ltd, pp. 953–964.

Sandberg, M. (1992), "VENTILATION EFFECTIVENESS AND PURGING FLOW RATE-A REVIEW", *International Symposium on Room Air Convection and Ventilation Effectiveness*.

Schulze, T., Gürlich, D. and Eicker, U. (2018), "Performance assessment of controlled natural ventilation for air quality control and passive cooling in existing and new office type buildings", *Energy and Buildings*, Elsevier Ltd, Vol. 172, pp. 265–278.

el Telbany, M.M.M., Mokhtarzadeh-Dehghan, M.R. and Reynolds, A.J. (1985), *Single-Sided Ventilation-Part I. The Flow between a Cavity and External Air Stream*, *Building and Environment*, Vol. 20.

Wang, H. and Chen, Q. (2015), "Modeling of the impact of different window types on single-sided natural ventilation", *Energy Procedia*, Vol. 78, Elsevier Ltd, pp. 1549–1555.

Wang, J., Wang, S., Zhang, T. and Battaglia, F. (2017), "Assessment of single-sided natural ventilation driven by buoyancy forces through variable window configurations", *Energy and Buildings*, Elsevier Ltd, Vol. 139, pp. 762–779.

Wei, Y., Guo-qiang, Z., Xiao, W., Jing, L. and San-xian, X. (2010a), "Potential model for single-sided naturally ventilated buildings in China", *Solar Energy*, Vol. 84 No. 9, pp. 1595–1600.

Wei, Y., Guo-qiang, Z., Xiao, W., Jing, L. and San-xian, X. (2010b), "Potential model for single-sided naturally ventilated buildings in China", *Solar Energy*, Vol. 84 No. 9, pp. 1595–1600.

Yakhot, V., Orszag, S.A., Thangam, S., Gatski, T.B. and Speziale, C.G. (1992), "Development of turbulence models for shear flows by a double expansion technique", *Physics of Fluids A: Fluid Dynamics*, Vol. 4 No. 7, pp. 1510–1520.

Zheng, J.W., Tao, Q.H. and Li, L. (2019), "Study of influence of shading louvers on wind characteristics around buildings under different wind directions", *IOP Conference Series: Earth and Environmental Science*, Vol. 238, Institute of Physics Publishing, available at:https://doi.org/10.1088/1755-1315/238/1/012031.