Wind Engineering Joint Usage/Research Center
FY2023 Research Result Report

Research Field: Outdoor Environment
Research Year: FY2023
Research Number: 23232010
Research Theme: Investigation of the influence of the eaves on the flow fields and wall pressure distribution of a building façade in a two-dimensional (2D) street canyon

Representative Researcher: Mohd Faizal Mohamad, Universiti Teknologi MARA (UiTM), Malaysia

Budget [FY2021]: Yen 350,000

*There is no limitation of the number of pages of this report.
*Figures can be included to the report, and they can also be colored.
*Submitted reports will be uploaded to the JURC Homepage.

1. Research Aim

The flow patterns in a two-dimensional (2D) street canyon are significantly affected by the physical characteristics of the buildings, such as the spacing between them, the shape of their roofs, and the design of their facades. Furthermore, the movement of air may only enter and exit the street canyon through the aperture located at the top of the structure. Altering the building's topography will impact the airflow within the street canyon, thereby influencing the ventilation of the buildings located on both the windward and leeward sides.
The aspect ratio, which refers to the distance between the buildings, is a commonly used measure for identifying the flow regimes of the street canyon. However, it is important to note that the eaves protruding beyond the façade also have a substantial impact on the turbulence structure (Alwi et al., 2023). Moreover, a quantitative investigation conducted by Ibrahim et al., (2023) has demonstrated that augmenting the length of the eaves leads to a higher scalar concentration, particularly on the side of the building that is sheltered from the wind. This particular architectural design is commonly observed in townhouses located in hot and humid regions of countries. These townhouses are arranged in a manner that is perfectly parallel to the roadway, resulting in the formation of a two-dimensional street canyon.
Although there have been many investigations conducted on the 2D street canyon, the majority of these studies have primarily examined the mean flow and turbulence characteristics within and above the canyon. The investigations are restricted to idealised building design that does not include any construction on the exterior walls. Furthermore, many studies have examined the impact of eaves on the alteration of flow and surface pressure of the structure, but these investigations have been restricted to the three-dimensional (3D) design of the building. Therefore, this study aims to clarify the ventilation mechanism of 2D street canyon. It intends to assess how the presence of protruding eaves affects the alteration of airflow within the canyon, using wind tunnel experiments. The data obtained from the experiment will serve as a benchmark for comparing a series of computational fluid dynamics (CFD) simulations. These simulations will involve different roof designs and eaves configurations.

2. Research Method

2.1 Building geometries

Figure 1 displays the schematic diagram of the buildings and the dimensions of the eaves that were employed in the current research. The buildings are characterised by dimensions of 320 mm in depth (x-axis), 2200 mm in length (y-axis), and 200 mm in height (z-axis). In this study, the building is assigned as H that is equal to 200 mm. Both buildings feature eaves on their facades that protrude in opposite directions, measuring 100 mm in length and 2 mm in thickness.

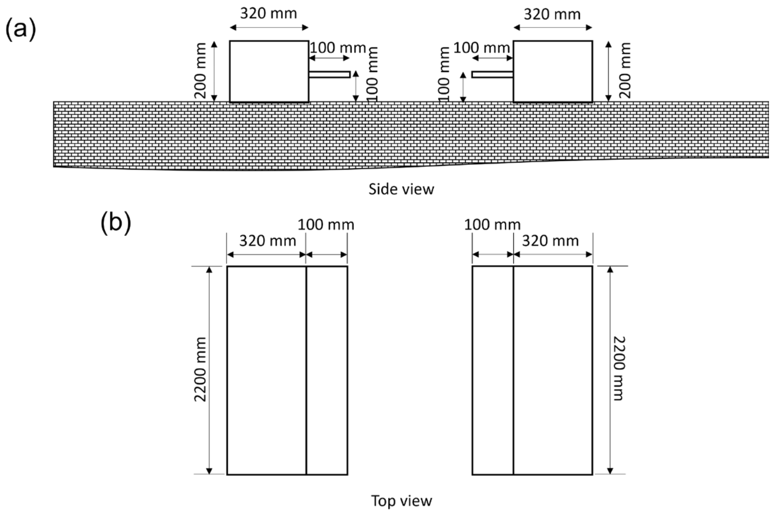

Fig. 1 The schematic representations of the buildings and eaves layouts. These representations were shown in both (a) side view and (b) top view.

2.2 Wind tunnel experiments

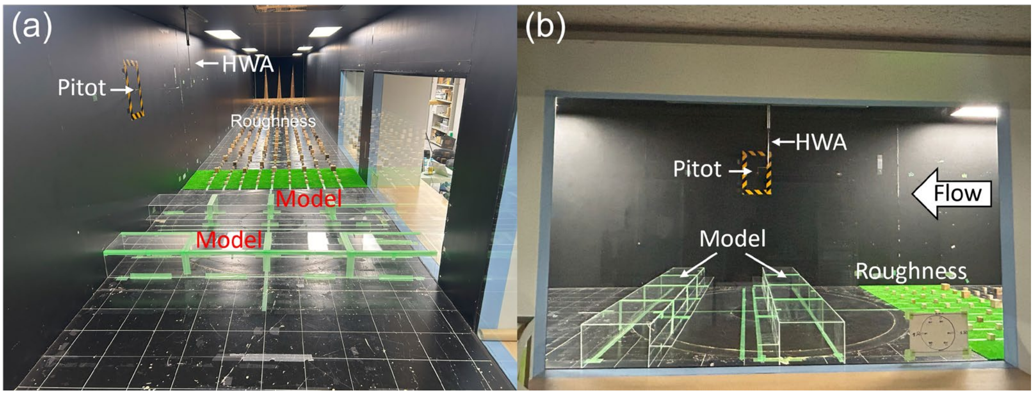

Fig. 2 Position of the street canyon in the wind tunnel with the position of pitot tube, hot wire anemometer (WHA) and roughness elements with flow from right to left (a) front view (b) side view.

The experiments were carried out in a wind tunnel at Tokyo Polytechnic University, Japan, which has a cross-section of 19 metres in length and 2.2 metres in width. Spires and roughness elements of 3 mm, 6 mm, and 9 mm were carefully positioned in the upstream area in a staggered arrangement, as depicted in Figure 2. The velocity profile is measured at a location $4H$ away from the upstream building façade. The measurements were conducted using an I-type hot-wire anemometer (HWA) with a sample time of 180 seconds.

2.3 CFD simulations

The Open Field Operation And Manipulation (OpenFOAM) software is used to conduct a series of computational fluid dynamic (CFD) simulations. The simulations solve the steady-state Reynolds-averaged Navier-Stokes (RANS) equations, and incorporate four (4) turbulence closure models: renormalization-group k-ε (RNG), standard k-ε (STD), realizable k-ε (RLZ), and shear stress transport k-ω (SST). The computational domain for the steady-state RANS simulations is depicted in Figure 3. The velocity profile obtained from the wind tunnel experiment is used as the inlet boundary conditions. The outlet is set zero static pressure, and the top surface of the domain exhibits slip boundary conditions. The building walls and ground surface are defined as boundaries with a no-slip condition. Figure 4 illustrates the variability of roof designs used in the CFD simulation. These designs are categorised as flat, wedge-1, pitched, and wedge-2, with eaves extending into the street canyon area. The gradient terms are subjected to second-order linear interpolation, while second-order discretization schemes are used to estimate the convection and viscous terms. The convergence is monitored by setting the residual values to 10^{-5} for pressure and 10^{-6} for all other equations.

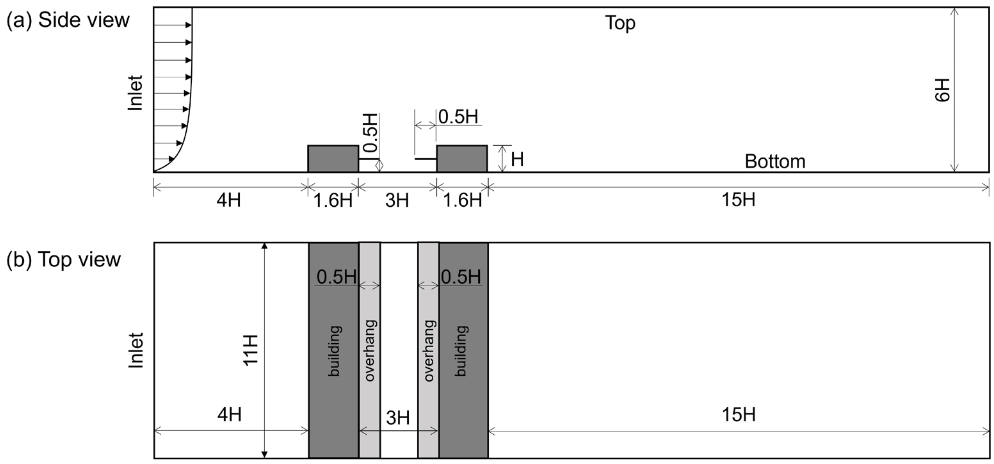

Fig. 3 Computational domain applied for the steady-state RANS simulations.

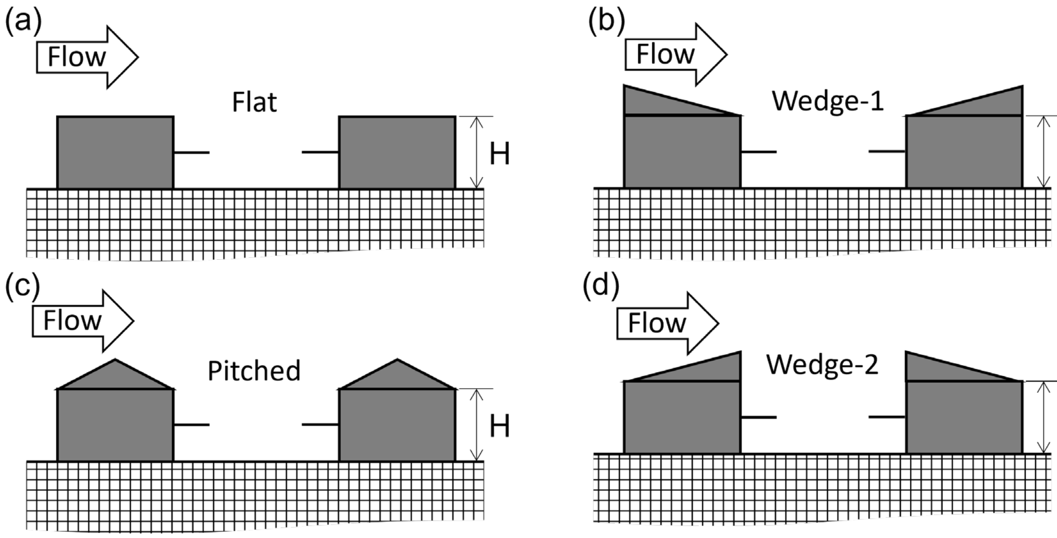

Fig. 4 Roof configurations adopted in the CFD simulations (a) flat (b) wedge-1 (c) pitched and (d) wedge-2.

3. Research Result

3.1 Velocity profiles

Figure 5 illustrates a comparison of velocity profiles between experimental results and predicted values from RNG, STD, RLZ, and SST models. The magnitude of two components, namely the streamwise velocity u and the vertical velocity w, is normalized with the reference velocity u_{ref}, which is measured at the height H of the building in the approaching flow. All models exhibit significant agreement with the experimental results at the location of L1 except near the wall where small discrepancies are observed. However, notable discrepancies are evident, particularly at the top section of the upstream building (L2~L4), where separation is the dominating phenomenon. Within the canyon, the STD model demonstrates satisfactory agreement with the experimental results by accurately predicting the flow patterns. However, in specific areas such as the lower section of L7 and L8, some underestimations can be observed. The remaining three models exhibit different trends in the region of $1.0 \leq z/H \leq 1.5$ for locations L5~L11. The STD model accurately predicts the velocity profiles on the top of the downstream building (L12 and L13) as compared to the experimental results.

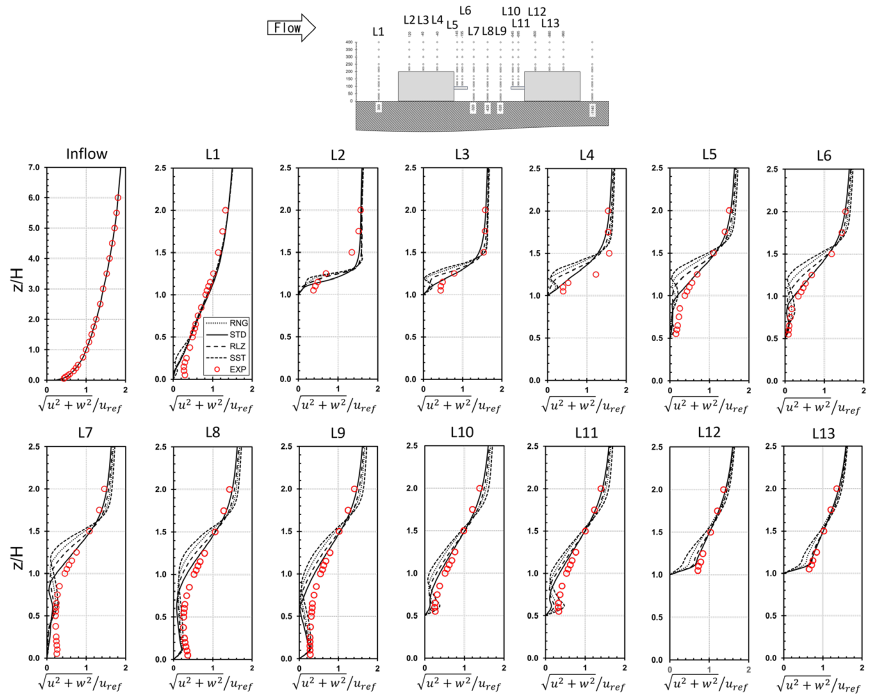

Fig. 5 Velocity profiles comparison of experiment (EXP), RNG, STD, RLZ and SST for different locations assigned as inflow and L1~L13.

3.2 Mean velocity contours and vectors

Figure 6 demonstrates the normalized streamwise velocity and velocity vectors for various roof configurations. In the case of a flat roof configuration (as shown in Figure 6 (a)), a nearly symmetrical recirculation region can be observed, with its center positioned approximately at the midpoint of the canyon height. The modification of the roof to a wedge-1 shape resulted in changes to the separation region at the edge of the upstream building, which improved the downward flow at the front façade of the downstream building. The presence of a pitched roof (as seen in Figure 6 (c)) amplifies the reverse flow within the canyon, in contrast to the flat roof condition. This serves as proof that the flow from above is being introduced into the canyon area. In addition, the wedge-2 shaped roof shifts the recirculation zone towards the downstream building by enhancing the downward flow along the building's edge.

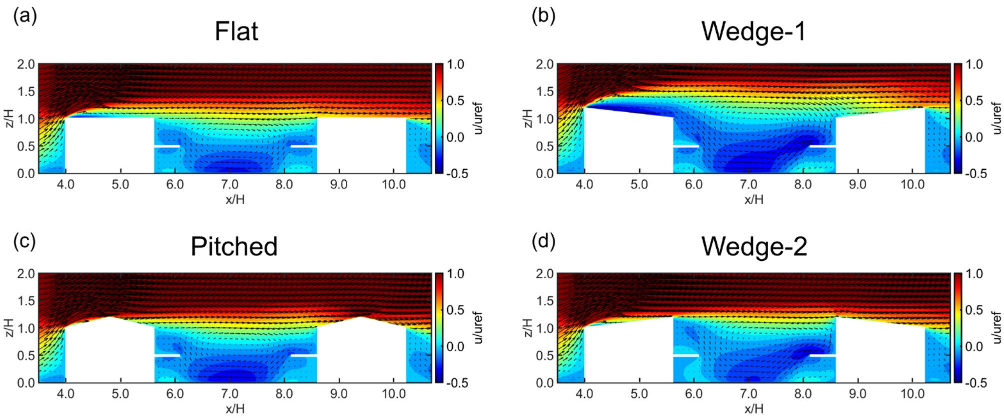

Figure 6 Normalized streamwise velocity contour and velocity vectors for (a) flat (b) wedge-1 (c) pitched and (d) wedge-2 roof configurations.

4. Published Paper etc.
[Underline the representative researcher and collaborate researchers]
[Published papers]
1. Not available
2. Not available

[Presentations at academic societies]
1. Not available
2. Not available

[Published books]
1. Not available
2. Not available

[Other]
Intellectual property rights, Homepage etc.

5. Research Group
1. Representative Researcher
Mohd Faizal Mohamad, Universiti Teknologi MARA (UiTM), Malaysia

2. Collaborate Researchers
1. Yoshie Ryuichiro, Tokyo Polytechnic University, Japan
2. Ikegaya Naoki, Kyushu University, Japan
3. Wei Wang, Kyushu University, Japan
4. Hirose Chiyoko, Fukuoka Institute of Health and Environmental Sciences, Japan
5. Azli Abd Razak, Universiti Teknologi MARA (UiTM), Malaysia

References

Alwi, A., Mohamad, M.F., Ikegaya, N., Razak, A.A., 2023. Effect of protruding eave on the turbulence structures over two-dimensional semi-open street canyon. Build. Environ.

228, 109921. https://doi.org/10.1016/j.buildenv.2022.109921
Ibrahim, M.F., Mohamad, M.F., Ikegaya, N., Razak, A.A., 2023. Numerical Investigation of Flow and Dispersion over Two-Dimensional Semi-Open Street Canyon. CFD Lett. 15, 53–70. https://doi.org/10.37934/cfdl.15.2.5370

6. Abstract (half page)

Research Theme: Investigation of the influence of the eaves on the flow fields and wall pressure distribution of a building façade in a two-dimensional (2D) street canyon
Representative Researcher (Affiliation): Mohd Faizal Mohamad, Universiti Teknologi MARA (UiTM), Malaysia
Summary・Figures
This study included performing a series of RANS simulations and comparing the findings with wind tunnel experiments. The simulations and experiments focused on a two-dimensional street canyon with eaves on both opposite facades and four different roof designs. The steady-state simulations are performed to assess the performance of various turbulence closure models (RNG, STD, RLZ, and SST). The major findings and future recommendations from this investigation can be briefly outlined as follows:

- The STD model outperforms the RNG, RLZ, and SST models in accurately predicting the velocity profiles at the upper part of the upstream building and within the street canyon. This discovery will provide valuable guidance for future research on the analysis of pollutant dispersion within the street canyon.
- Various roof forms have significant effects on the flow structure within a street canyon. Further work is needed to assess the ventilation effectiveness within the canyon, as the presence of eaves hinders the flow in the respective area.