

Performance evaluation of RANS models for cross ventilation using LES and PIV as benchmark data

Kainan Liao ^a, Wei Wang ^b, Naoki Ikegaya ^c

^aInterdisciplinary Graduate School of Engineering Sciences, Kyushu University, Fukuoka, Japan, liao.kai.nan.528@s.kyushu-u.ac.jp

^bFaculty of Engineering Sciences, Kyushu University, Fukuoka, Japan, wang.wei.530@m.kyushu-u.ac.jp

^cFaculty of Engineering Sciences, Kyushu University, Fukuoka, Japan, ikegaya.naoki.116@m.kyushu-u.ac.jp

SUMMARY

Turbulent wind-driven ventilation is crucial for indoor air renewal when opening faces are parallel to the dominant wind direction. Although numerical models have been employed for various ventilation studies, this study systematically assesses the standard k - ϵ Reynolds-averaged Navier–Stokes (RANS) model against large-eddy simulation (LES) and high-resolution particle image velocimetry (PIV) measurements for a ventilation scenario where turbulent air exchange predominantly determines the indoor velocity distribution and flow rate. The PIV dataset, partially employing the stereoscopic method, provides the three velocity components within the opening, enabling the direct determination of the airflow rate. Comparisons focus on mean and turbulent flow structures within the model as well as the flow distribution and exchange rate on the window face. The RANS results lack the precision in predicting reverse flow regions and shear-induced turbulence, yielding to inaccurate inlet/outlet regions at the opening. In contrast, LES demonstrates superiority in resolving these turbulence-driven phenomena.

Keywords: Wind-driven turbulent ventilation, Reynolds-averaged Navier–Stokes simulation, Large-eddy simulation, Particle image velocimetry

1. INTRODUCTION

Effective wind-driven cross ventilation is essential for reducing building energy consumption and maintaining acceptable indoor air quality. However, accurately modeling the indoor airflow field remains a major challenge in computational fluid dynamics (CFD), particularly because the exchange through openings is governed by unsteady, turbulence-driven mechanisms rather than steady pressure differences when opening faces are parallel to the dominant wind direction. Despite this, most previous CFD studies have not examined how well steady Reynolds-averaged Navier–Stokes simulation (RANS) models capture the indoor flow structures and turbulence characteristics that determine ventilation performance for such scenarios, leaving a critical gap in current knowledge. High-fidelity large-eddy simulation (LES) can overcome many of these deficiencies by better reproducing near-opening dynamics and indoor velocity distributions, as demonstrated in prior experimental validation studies (Miura et al., 2025). However, its high computational cost and time are often prohibitive for routine design and parametric analysis (Jiang et al., 2004). RANS simulation, particularly the standard k - ϵ model, has been employed and even demanded for engineering purposes due to their computational efficiency, although the model is known to have certain limitations in prediction accuracy for various scenarios in built-environment (e.g., impinging flow to a building face). To quantify the applicability of the RANS model for more complex ventilation cases, the present study conducts a systematic assessment of the standard k - ϵ model by coupling indoor and outdoor airflow analyses. The validation of RANS predictions, particularly in regard to mean flow structures, turbulence intensity, and indoor circulation patterns, is conducted by comparing them to high-fidelity LES results. The particle image velocimetry (PIV)

experimental data, partially employing the stereoscopic method, serves as the definitive benchmark in this process. By maintaining consistent boundary conditions and geometry across all methods, this study isolates the fundamental advantages and disadvantages of steady RANS simulation in reproducing realistic ventilation phenomena.

2. METHODOLOGY

Fig. 1 (b) is a cubic building ($H = 0.32$ m) with two opposite openings ($L \times L = 0.1 \times 0.1$ m²) and a wall thickness of 0.002 m, following the geometry used in the two-dimensional two-velocity and three-velocity component (2D2C and 2D3D) PIV measurements by Wang et al. (2025). As shown in the domain overview in Fig. 1 (a), the computational domain dimensions were established as 6.12 m, 2.2 m, 1.8 m in the streamwise, spanwise, and vertical directions, respectively. The corresponding coordinates and velocity components are denoted as x, y, z and u, v, w . To resolve the flow field accurately, the domain was discretized by approximately 9.7 million hexahedral orthogonal cells. Local refinement was applied around the building and window openings to capture flow separation and high-gradient regions. Specifically, the minimum cell size was set to 0.001 m at the building corners to resolve the thin walls, with a constant growth ratio of 1.08 applied to cells expanding away from the building surfaces.

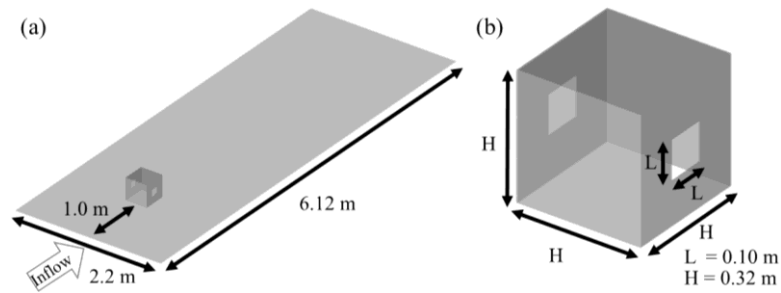


Fig. 1. Computational domain and geometry: (a) overview of the domain dimensions and placement. (b) close-up view of the cubic building model ($H = 0.32$ m) with two opposite openings ($L \times L = 0.1 \times 0.1$ m²).

Numerical simulations were performed using OpenFOAM v2106 (OpenCFD Ltd.). The standard $k-\epsilon$ model was applied. The semi-implicit method for pressure linked equations (SIMPLE) algorithm was utilized to handle the pressure-velocity coupling. For spatial discretization, the second-order linear-interpolation scheme was used for gradient terms, while a second-order upwind scheme (linearUpwind) was employed for divergence terms to ensure numerical stability. Boundary conditions were established to match the wind tunnel experiment. The inlet prescribed a power-law velocity profile consistent with the experimental data. A zero-pressure condition was applied at the outlet. For the side, top, ground and building surfaces, the Spalding's wall function was utilized. For comparison, a reference LES (based on the standard Smagorinsky model using the identical numerical mesh) was conducted under identical conditions to study the influence of turbulence modeling (Miura et al., 2025).

3. RESULTS AND DISCUSSION

Fig. 2 compares the distributions of the mean velocity component \bar{v}/u_H and the Reynolds normal stress σ_v^2/u_H^2 at the indoor mid-height cross-section ($z/H = 0.5$). It should be noted that the red-boxed regions in the WTE figures (Figs. 2(c, f)) signify areas where measurement accuracy is

compromised. Consequently, these specific regions must be excluded from the validation process. Here, the overbar indicates the time averaged quantity. u_H represents the mean wind speed at $z = H$ of the approaching flow. The values of σ_v^2/u_H^2 for the RANS were calculated based on the flux-gradient formulation using the turbulent kinematic viscosity $\nu_t = C_\mu k^2/\varepsilon$, where k and ε are the turbulent kinetic energy and dissipation rate, $C_\mu = 0.09$ is the model constant in the k - ε model. In contrast, σ_v^2/u_H^2 in the LES and PIV represent the statistics directly calculated from the obtained time-series data of v . In Figs. 2(a, b, c) for \bar{v}/u_H , the LES results exhibit a highly similar distribution pattern to the WTE data, showing the reverse flow regions in the LES and WTE are comparable in size and structure. In contrast, the RANS model in Fig. 2(a) predicts a noticeably smaller reverse flow region. Furthermore, the locations of the incoming ($\bar{v} < 0$) and outgoing ($\bar{v} > 0$) flows within the opening differ distinctly from those observed in both LES and WTE. These discrepancies are likely caused by the weaker capability of RANS to predict flow separation accurately. The relatively low accuracy of RANS in reproducing reverse flow has been similarly reported in previous outdoor flow simulations. A similar trend is observed in the Reynolds stress distributions in Figs. 2(d, e, f). The LES prediction (Fig. 2(e)) captures the spatial distribution of turbulence more accurately than RANS (Fig. 2(d)) when compared to the experimental benchmark (Fig. 2(f)).

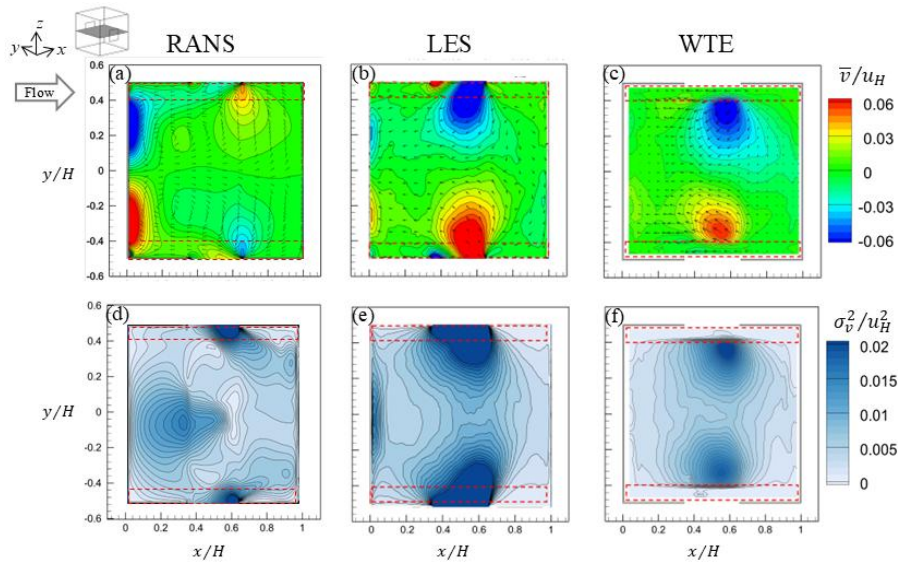


Fig. 2. Comparison of (a, b, c) mean velocity and (d, e, f) turbulence statistics at the indoor mid-height horizontal plane ($z/H = 0.5$) obtained from (a, d) RANS, (b, e) LES, and (c, f) WTE.

Fig. 3 compares the flow characteristics within the opening vertical section described by \bar{v}/u_H and σ_v^2/u_H^2 . These quantities reveal the inflow behavior and turbulence generation associated with the shear layer that develops around the opening. Regarding the mean vertical velocity in Figs. 3(a, b, c), the LES accurately reproduces the WTE features, as both methods show very similar airflow patterns for the entering and exiting flow. In contrast, while the RANS prediction captures the general ventilation flow tendency that inflow at the bottom-right and outflow at the top-left, the prediction is not precise. The distribution in the RANS model appears overly smooth and lacks the distinct vertical variations observed in both the LES and WTE data. A similar trend is evident in the turbulence-related quantities shown in Figs. 3(d, e, f). The LES prediction is highly consistent

with the WTE data, successfully capturing the intensified turbulence along the window perimeter caused by shear at the opening. However, the RANS results show notable deviations. The variance is significantly underpredicted along both the windward and leeward edges, indicating that RANS does not adequately capture the shear-induced turbulence development. Consequently, the mean velocity and turbulence profiles within the RANS results are overly diffused compared to the more accurate LES and WTE results.

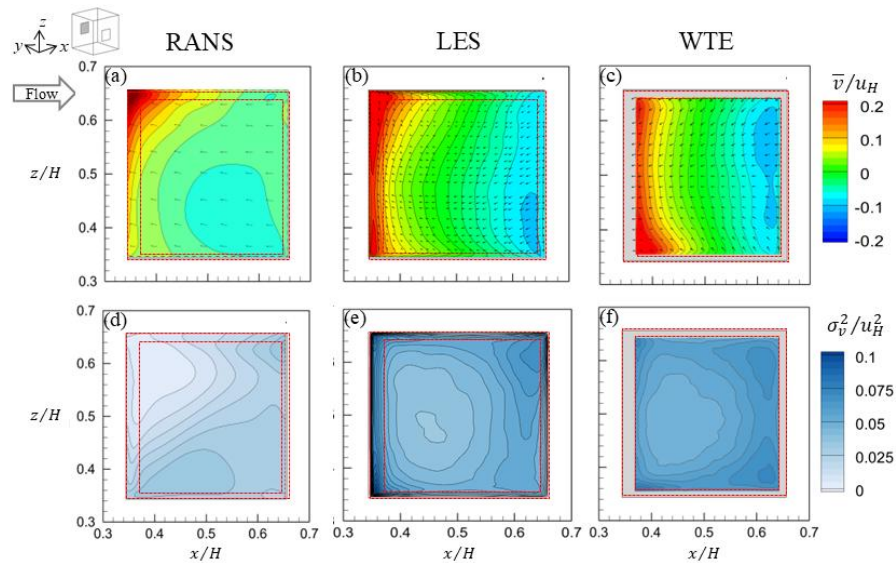


Fig. 3. Comparison of (a, b, c) mean velocity and (d, e, f) turbulence statistics at the vertical window plane obtained from (a, d) RANS, (b, e) LES, and (c, f) WTE

4. CONCLUSION

The present study evaluated the performance of RANS (the standard $k-\epsilon$) and LES (the standard Smagorinsky) models for turbulence-dominated lateral cross-ventilation against wind tunnel benchmarks. The findings indicate that LES exhibits superior fidelity, accurately reproducing complex reverse flow structures and shear-induced turbulence. Conversely, RANS captures only general mean-flow trends and lacks precision, notably predicting smaller reverse flow regions and inaccurate inlet/outlet locations. Furthermore, the RANS method yields profiles that are excessively smoothed, and it significantly underpredicts turbulence in shear layers. Consequently, given the computational time and cost considerations, RANS should be prioritized for rapid preliminary screening, whereas LES is recommended for high-fidelity analysis requiring detailed physical insights.

REFERENCES

- Miura, G., Wang, W., & Ikegaya, N. (2025). Computational Fluid Dynamics Simulation for Determining Unsteady Ventilation Rate from Side Openings. Summaries to Technical Papers of Annual Meeting, Japan Association for Wind Engineering, 2025, 149. https://doi.org/https://doi.org/10.14887/jaweam.2025.0_149
- Wang, W., Hirose, C., & Ikegaya, N. (2025). Directly determining mean and instantaneous ventilation rates using stereoscopic PIV for single-sided and cross ventilations of a cubic building. *Sustainable Cities and Society*, 118, 106049. <https://doi.org/10.1016/J.SCS.2024.106049>
- Jiang, Y., Allocca, C., & Chen, Q. (2004). Validation of CFD Simulations for Natural Ventilation. *International Journal of Ventilation*, 2(4), 359–369. <https://doi.org/10.1080/14733315.2004.11683678>
- OpenCFD, (n.d.). <https://www.openfoam.com/news/main-news/openfoam-v2106>.