NUMERICAL SIMULATION OF STREET-LEVEL WINDS WITHIN A PASSAGE USING A TWO-LAYER \( k-c \) MODEL

Cheng-Hu HU\(^1\), Nobuyuki KOYAYASHI\(^2\) and Masaaki OHBA\(^3\)

ABSTRACT

The flow field in the passage between two buildings is simulated using a two-layer \( k-c \) model and the results are compared with the wind tunnel data. This study concentrates on the variations of the mean wind speeds within the passage under different wind directions (0°, 15°, 30°, 45°, 60°, 90°) and the influence of the height difference between the two buildings is also investigated. Good agreement is obtained when the two buildings are of the same height. Although the agreement is less satisfactory when there is a height difference between the two buildings, the general trend of the wind velocity variations within the passage is similar to that of the experiment. The salient flow features can still be depicted by this economical approach and it is considered adequate at the stages of preliminary design.

Key Words: Street-level wind, Two-layer \( k-c \) model, Wind environment

1. INTRODUCTION

Street-level winds are one of the primary concerns in environmental design. Architects and urban planners need to mitigate unfavourable or dangerous winds that may occur in the areas accessible to pedestrians, since "wind nuisance" may impair some desired functions of the outdoor spaces. Using CFD to simulate the street-level winds has gradually been a common practice among building professionals at the stages of preliminary design. Currently the standard \( k-c \) model is still widely used since it is economical and stable in computation. However, the standard \( k-c \) model requires wall functions to resolve the boundary layer but those wall functions are not ideal for separated flows. Low-Reynolds number \( k-c \) models neglect the use of wall functions but they usually render difficulties in computation due to the requirement of very fine grids, since the gradient of \( c \) is very steep near the walls. The two-layer \( k-c \) model does not employ wall functions and the grids near the walls need not be very fine. Therefore it may improve the performance of the standard \( k-c \) model in the simulation of airflow around buildings. This study reports an application of the two-layer model and the comparisons with experimental data for discussions.

2. GOVERNING EQUATIONS

1) COE Researcher, 2) President, Tokyo Polytechnic University, 3) Professor, Department of Architecture, Tokyo Polytechnic University, 1583 Iiyama, Atsugi, Kanagawa 243-0297
The two-layer $k$-$\epsilon$ model applied in this computation was of Rodi [1]. This model uses the standard $k$-$\epsilon$ model in the outer layer, whereas a prescribed length scale is employed to resolve the near wall layer (inner layer), as follows:

$$
\nu_t = C_\mu k^{1/2} l_\mu; \quad l_\mu = C_{\lambda \mu} y_n \left( 1 - \exp \left( -\frac{\text{Re}_y}{A_\mu A'} \right) \right) \quad ; \quad C_{\lambda} = \kappa C_{\mu}^{-3/4} \quad ; \quad \text{Re}_y = \frac{k^{1/2} y_n}{\nu}
$$

(1)

where $\nu_t$ is the eddy viscosity, $k$ the turbulent kinetic energy, $y_n$ the distance to the nearest wall, $\kappa$ the von Kármán constant ($-0.41$), $C_{\mu} = 0.09$, $A_\mu = 50.5$ and $A' = 25$. The prescribed length scale $l_\mu$ is controlled by the damping function in which the local Reynolds number ($\text{Re}_y$) is a variable. Since $\text{Re}_y$ does not involve the friction velocity $u'$, the length scale prescription defined in equation (1) is considered also applicable in separated flows. The distribution of $\epsilon$ in the inner layer is calculated from

$$
\epsilon = \frac{k^{2/3}}{l_\epsilon}; \quad l_\epsilon = \frac{C_{\lambda \epsilon} y_n}{1 + 5.3 / \text{Re}_y}
$$

(2)

which is an algebraic equation. This can improve convergence in computation and the demand of very fine mesh in the near wall regions can also be relaxed. The inner layer and the outer layer is matched at $\text{Re}_y = 350$ where the viscous effects due to the presence of wall is negligible. The turbulence model closes the Reynolds Averaged Navier-Stoke (RANS) Equations. The RANS equations and the turbulence model were discretised by the 3rd-order “SMART” differencing scheme [2] to determine the cell-face values of the velocity variables ($U, V, W$) as well as $k$ and $\epsilon$. A commercially available CFD code “PHOENICS” [3] was used to solve these equations.

3. MODELLING APPROACH

The wind tunnel data is obtained from Stathopoulos and Storms [4] in which they use a TSI hot-film anemometer to measure the variations of the wind velocities within the passage. To model the flow field, the size of the computing domain was $1.8 \text{ m} \times 1.8 \text{ m} \times 1.0 \text{ m}$ and the number of grids was $81(x) \times 97(y) \times 43(z)$. The mean wind velocity profile was defined by a power-law model with an exponent of 0.15. The $k$ was obtained from $(U(z) \times I(z))^2$, where $U(z)$ is the mean wind velocity and $I(z)$ is the turbulence intensity. The profile of $\epsilon$ was defined by $C_{\mu}^{0.75} k^{1/3} (kz)$. The model buildings were scaled to $1/400$, which was the same as the scaled model in the wind tunnel. The $k$ was fixed to zero as the boundary condition at the walls and the velocity variables parallel to the walls in the near wall regions were assumed to follow the log-law profile. The flow field of interests was the street-level winds within the passage formed by the two buildings (figure 1).

Figure 1 The geometry and building layout. Dimensions shown are in full scale whilst the model used is scaled. The investigation is made along the passage centreline.
Different wind angles (0°, 15°, 30°, 45°, 60°, 90°) were simulated and the simulated wind fields were applied in the cases when the two buildings were of the same height (H : H) and when one of the buildings was three times taller than the other (3H : H). Computational grids were of the Cartesian type and the nearest nodes adjacent to the walls were in the range of 10 < y^+ < 30 in order to comply with the resolution requirement of the two-layer model.

4. RESULTS

The locations (in full scale) measured were at 0 m, 4 m, 8 m, 12 m, 16 m and 20 m along the centreline of the passage and the measuring height was 2 m above ground. The measured wind speed was \( Q = (U^2 + U'V')^{1/2} \), since the orientation of the probe was horizontal; therefore the detected velocity components were effectively the streamwise (U) and the vertical (V') quantities. The measured wind speeds were normalised by a reference wind velocity (\( V_r \)) which was measured at each corresponding location when the buildings were absent. With the same manner, the computed wind velocities were also expressed by the values of \( Q \) and the computed values were all normalised by their corresponding reference wind velocities. The first set of data for comparison is the cases of the two buildings were of the same height. In figure 2, the experimental data indicate that the wind speed is accelerated near the entrance and gradually decreases till the exit in the two occasions (\( \alpha = 0^\circ \) and 15°). The amplification of wind speed is about 40% at the location around 4 m and it decreases steadily to approximately equal to the reference wind speed at the exit. The computed values depicted the same trend and the velocity ratios were very close to the measured values with a discrepancy in a range within 10%.

![Figure 2 Comparison of velocity ratios and the simulated wind fields (left: \( \alpha = 0^\circ \) and right: \( \alpha = 15^\circ \) )](image)

When the wind angle is 30°, the maximum amplification of the wind speed indicated by the experimental data is also about 40% and the location where the wind is most likely to be accelerated is still around 4 m from the entrance (figure 3). In figure 3, it also shows that in the case when the wind angle is 45°, a rather different trend of the wind velocity distribution is present. The experimental data indicate that the wind velocities are slightly lower than 1.0 around the halfway of the passage and the maximum amplification of the wind speed decreases to about 20% at 4 m from the entrance. The computed values were still in good agreement with the experimental data but the agreement was slightly less satisfactory when the wind angle was 45°, in which case the maximum discrepancy was 20%.

When the wind angle increases to 60°, the experimental data shows that most of the velocity ratios are lower than 1.0 except the measured value at the entrance (figure 4). The computed values agreed well with the data in most of the locations though, the agreement was less satisfactory at the entrance of the passage. However, the velocity ratio at the entrance under this wind direction was likely to be lower than 1.0 because of the shelter effects. The maximum
discrepancy was 25% in this occasion. Increasing the azimuth to 90°, the wind angle becomes orthogonal to the passage. The measured values indicate that the maximum velocity ratio is about 20% of the undisturbed wind speed. The computed values predicted the same trend and the agreement between computation and experiment was fairly well.

Figure 3 Comparison of velocity ratios and the simulated wind fields (left: α = 30° and right: α = 45°)

Figure 4 Comparison of velocity ratios (left: α = 60° and right: α = 90°)

Figure 5 Comparison of velocity ratios when H : H = 3 : 1 (left: α = 0° and right: α = 15°)

In the cases when one of the buildings is three times taller than the other, the experimental data shows that the wind velocities have been largely accelerated due to the height difference and the maximum amplification of wind
speed is 60% when the wind direction is parallel with the passage. The computed values were lower than the experimental data and the error percentage was between 11% and 26% (figure 5). A possible explanation was that there might be stronger downwash and reverse or recirculating flows in that region because the hot-film probe was likely to report a higher velocity if the reverse flows were strong enough to make contributions.

When the azimuth = 15°, the experimental data shows that the maximum amplification of wind speed is 55% at a distance around 5 m from the entrance. The computation predicted a similar trend with small discrepancies. The error percentage was in a range between 7% and 14%. The predicted values agreed favourably with the measured values.

![Graph](image)

**Figure 6** Comparison of velocity ratios when H : H = 3 : 1 (left: α = 30° and right: α = 45°)

In the cases when the azimuth = 30° and 45° (figure 6), the experimental data indicate that most of the velocity ratios are still greater than 1.0, implying the tall building can still cause acceleration of wind speed even when the wind direction is relatively oblique to the passage. The amplifications of wind speeds became lower due to the weakened flows in the wake but the strength of the downwash flows from the tall building was likely to increase. The computations predicted similar trends in both cases but the agreement was less satisfactory in the range between 0 m and 10 m from the entrance of the passage. The maximum discrepancy was 20% in the case when the wind angle was 30° and it increased to 32% when the wind direction was 45° obliquely to the passage. Figure 7 shows the velocity vectors in the horizontal and vertical planes.

![Graph](image)

**Figure 7** Velocity vectors in the horizontal plane (left) and in the vertical plane (right) when α = 45°

In the horizontal plane, the velocity vectors were amplified at the entrance (distance = 0 m) but the magnification was mainly on the \( V \) component; therefore it rendered few contributions to the \( U \) component. The velocity vectors
gradually changed their directions between 0 m and 10 m from the entrance and these vectors became fully parallel to the passage after a distance of 10 m. Therefore, the $U$ component was dominant in the wind velocity after that distance. In the vertical plane, there were some upward flows near the entrance, indicating the downwash and recirculating flows were likely to present in that region. This might cause some difficulties in the measurement and it was also likely to downgrade the performance of the turbulence model due to highly complicated flow patterns.

![Figure 8](image_url)

Figure 8: Comparison of velocity ratios when $H : H = 3 : 1$ (left: $\alpha = 60^\circ$ and right: $\alpha = 90^\circ$)

When the wind angle becomes $60^\circ$, the experimental data indicate that the wind velocity is as strong as the reference wind speed and it decreases gradually and reaches a lowest value at the location around 8 m from the entrance and, subsequently the wind velocity increases more rapidly till the exit of the passage (figure 8). The computation depicted the same trend except at the location near the entrance. The maximum discrepancy was 46% in this case. The reason could also be explained using figure 7 in the same manner; the $U$ component was less amplified at the entrance and the flow vectors were complicated and the wind velocities detected by the probe were likely to be higher than the actual wind speeds in the vicinity of the entrance due to recirculating flows. When the wind direction is orthogonal to the passage, the experimental data indicate that the wind speed is about 50% of the reference wind velocity in the middle of the passage and the velocity ratio increases to 1.08 at the locations near the entrance and the exit. The computation predicted a same trend with an error percentage between 12% and 41%. The larger discrepancy found in this case was also likely due to the strong downwash flows created in this wind direction.

5. CONCLUSIONS

Wind field around buildings is highly complicated and it is more difficult to resolve if there are several buildings involved. The discrepancies found in this study can be attributed to the simplified assumptions embedded in the turbulence model and some inadequacies in instrumentation. Use of the two-layer $k$-$\varepsilon$ model was not completely satisfactory but it was adequate to resolve salient flow features of concerns at the stages of preliminary design.

REFERENCES