A Wind Tunnel Full-Scale Building Model Comparison between Experimental and CFD Results Based on the Standard \( k-\varepsilon \) Turbulence Representation

Shigeki Nishizawa\(^1\), Takao Sawachi\(^1\), Ken-ichi Narita\(^2\), Hironao Seto\(^1\) and Yumi Ishikawa\(^1\)

\(^1\) Building Research Institute, Japan
\(^2\) Nippon Institute of Technology, Japan

Abstract

To evaluate the property of cross ventilation quantitatively, it is important that the calculated air flow field is compared with measurement. In this paper, the air flow field in the wind tunnel of the Building Research Institute of Japan (BRI) was calculated by CFD analysis using the standard \( k-\varepsilon \) model, and the adequacy of the calculation was examined by comparison with measured values. Results showed that:

- The calculated air flow field was generally in good agreement with the measured field;
- The distribution of the wind pressure coefficient was similar between measurement and calculation;
- The calculated value of wind pressure coefficient was lower than measurement;
- The turbulent kinetic energy was not significantly overestimated;
- The differential pressure between openings showed a good relation to measurement;
- The calculated indoor air flow, inside a simple building model enclosed in the wind tunnel, was strongly influenced by external conditions.

Key words: Cross ventilation, wind tunnel experiment, standard \( k-\varepsilon \) model, CFD.

1. Introduction

Cross ventilation has been used successfully in Japan to control the indoor environment in summer. However, the number of buildings suitable for cross ventilation has decreased as a result of the development of air conditioners combined with a deterioration of the external environment and a changing pattern of lifestyle. Nevertheless, cross ventilation remains one of the most important techniques for achieving energy conservation and for maintaining a comfortable indoor environment during summer in temperate regions. It is comparatively easy to develop a qualitative design for the cross ventilation of a space with the easiest approach being to set wide openings for the prevailing wind direction. However, it is difficult to evaluate the effect of cross ventilation quantitatively and to design the ventilation of a space based on a quantitative evaluation. This is because the indoor environment changes greatly with time as a result of changing wind direction and thermal conditions. To evaluate the property of cross ventilation quantitatively, wind tunnel experiments and numerical simulation (Computational Fluid Dynamics: CFD) have been performed. Increasingly, CFD is used to design cross ventilation and many commercial codes have been developed. There are, however, various issues to consider including choice of turbulence model (typically standard and modified \( k-\varepsilon \) are incorporated in commercial code), the setting of boundary conditions, computational capacity and processing time etc. Although CFD is widely used, validity is not certain (especially, when using the standard \( k-\varepsilon \) model). It is important therefore to compare calculation with experimental measurement. At the Building Research Institute in Japan, a wind tunnel incorporating a full-scale building model has been developed. Using this, the air flow field characteristics have been measured in detail (Sawachi et al 1999 and 2004). In this paper, the measured air flow field in the wind tunnel has been compared with that calculated by CFD analysis using the standard \( k-\varepsilon \) turbulence model. From this, the adequacy of calculation has been examined.

2. Outline of Experimental Measurement and Simulation

The plan and section of the BRI wind tunnel is shown in Figure 1. This wind tunnel is configured to examine the property of air flow in and around a full-scale building model. Its form is different to a
conventional boundary layer wind tunnel in that it provides a uniform flow pattern. Also, a large size full-scale building model was used, resulting in a large blockage ratio of 12%. In addition, the distance leading to the building is short and the cross-sectional area of the air inlet to the working section (W=9,500mm, H=5,000mm) is smaller than the cross-sectional area of the working section itself (W=15,500mm, H=9,000mm). These properties greatly influence the air flow pattern and create a different pattern than would be experienced in a conventional boundary layer wind tunnel. In addition, turbulence is reduced by incorporating a honeycomb baffle on the inlet side.

A plan view of the building model, inserted in the wind tunnel, is illustrated in Figure 2. It has dimensions W=D=5,560mm and H=3,000mm and has four rooms. Set close to the diagonal are two large openings of size W=860mm, H=1,740mm. The effect of wind direction was analysed by rotating the model in 15 degree intervals.

In the experimental measurement, the velocity in and around the building, and the static pressure on the wall, roof and floor were measured. Both a coarse and a fine mesh of measurement points for velocity and pressure were applied as shown in Figures 2 and 3 respectively. Measurement points of static pressure were located at 60 points on each wall (Figure 4), and at 144 points on the roof. Velocity was measured using a 3-dimensional ultrasonic anemometer (Kaijo, WA-390) while static pressure was measured using a differential manometer (Baratron Type 220C). The static pressure of a pitot tube at 4,500mm in height, near the air inlet to the working section (Figure 1), was used as the standard pressure of the differential manometer.

The velocity at the air inlet to the working section was set at “3m/s”. However this velocity is not uniform over the area of the air inlet, because the building model itself influences the air flow on the windward side. Thus this velocity of “3m/s” represents an area-average value of the inlet value.

The commercial CFD program used in the analysis was STREAM for Windows V4 (Software Cradle). This software is widely used in Japan. An outline of the settings used in STREAM is shown in Table 1. For this study, the calculation area represented the working section of the wind tunnel as shown in Figure 1. The mean velocity at the inflow boundary (Section A-A’ in Figure 1), $U_{in}$, was set as shown in Figure 5a; $U_{in}$ is not uniform along section A-A’.

![Figure 1. Section and plan of the BRI wind tunnel.](image1)

![Figure 2. Large (coarse) grid around the model.](image2)

![Figure 3. Small (fine) grid used within and adjacent](image3)
Table 1. Outline of CFD analysis.

<table>
<thead>
<tr>
<th>Software Name</th>
<th>STREAM for Windows V4 (Software Cradle)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Turbulent Model</td>
<td>Standard k-ε Model</td>
</tr>
<tr>
<td>Algorithm</td>
<td>SIMPLEC Algorithm</td>
</tr>
<tr>
<td>Difference Scheme</td>
<td>QUICK</td>
</tr>
<tr>
<td>Inflow Boundary</td>
<td>Setting Uin</td>
</tr>
<tr>
<td>Outflow Boundary</td>
<td>Setting Mass Flow</td>
</tr>
<tr>
<td>Wall Surface</td>
<td>Standard Log-Law</td>
</tr>
<tr>
<td>Mesh</td>
<td>120(X)*120(Y)*63(Z)</td>
</tr>
</tbody>
</table>

**Wind Direction**

<table>
<thead>
<tr>
<th>Wind Direction</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>0° - 45°</td>
<td>(without Openings)</td>
</tr>
<tr>
<td>0° - 165°</td>
<td>(with Openings)</td>
</tr>
</tbody>
</table>

**Figure 4. Elevation from windward, 0° wind direction.**

**Figure 5 Decision of U_in**

The turbulent kinetic energy, $k_{\infty}$, and dissipation rate, $e_{\infty}$, at the inflow boundary are set to the given default value in STREAM of $10^{-10}$. This low value is used because the honeycomb structure of the inlet is designed to minimize turbulence. As previously described, the turbulence model was the standard k-ε model and the wall boundary was represented by the Log-Law. This combination is often used for simulation of cross ventilation because of public recognition and simplicity. An intention of this paper, therefore, was to examine this popular condition when using commercial code.

Wind directions of $0°$-$45°$ were applied under conditions without openings (i.e. a solid model), and $0°$-$165°$ under the condition of large openings set on the diagonal (Figure 3). The condition without openings is set to evaluate air flow around the building model and the static pressure on the surface of the building. The condition of large openings was set mainly to evaluate the indoor air flow and ventilation rate.

3. Results and Discussion

3.1 Air Flow around the Building Model

The calculation and experimental results for flow without openings (wind direction: $0°$, $45°$) in the building are shown in Figures 6 to 15. Under this condition the property of air flow around the building model was examined.

Figures 6 and 7 show the root-mean-square (rms) of the turbulent velocity fluctuations and mean velocity in section (for $0°$ wind direction and no openings) where the rms of the turbulent velocity fluctuations is defined as:

$$\sqrt{u'^2} = \sqrt{u^2 + v^2 + w^2} = \sqrt{2k}$$  \hspace{1cm} (1)

where $u$, $v$, $w$ are turbulent velocity components, and $k$ is the turbulent kinetic energy.

Generally, it is pointed out that the standard k-ε model has some problems, the most common being that the calculated turbulent kinetic energy, $k$, is overestimated in the collision area and the calculated distribution of wind pressure coefficient, $C_p$, on the windward side is different from measurement. In the comparisons undertaken in this research, simulated rms values of the turbulent velocity fluctuations were found to be lower at the location where the air stream meets the windward facing wall (Figure 6b location A).
Figure 6. RMS of turbulent velocity, $\sqrt{\bar{u}^2}$, (0°, no opening, Section C-C').

Figure 7. Mean velocity distribution (0°, no opening, Section C-C').

Figure 8. Wind pressure coefficient (0°, no opening).

Figure 9. Mean velocity (0°, no opening, Z=1000 mm).
Results also showed that the observed horseshoe vortex was appropriately expressed (Figure 6b location B) and the calculated mean velocity field is similar to the experimental field (Figures 7 and 9). There are, however, some differences in the wake (Figures 6 and 7 location C) and at the edges of the main stream (Figure 9 locations D). The distribution of the wind pressure coefficient on the windward side is also similar, although the calculated value is about 80% of the measurement value (Figure 8). In addition, the maximum wind pressure coefficient is greater than 1.0 but this is because the total pressure at the pitot tube (Figure 1) refers to the condition without the building model being present.

The turbulent kinetic energy was kept lower level in the upwind building boundary region than the evaluation of previously stated studies using the standard k-ε model. This is because of the inflow boundary condition and the form of the wind tunnel. The inlet, incorporating the honeycomb baffle, results in low turbulent kinetic energy of the inflowing air. Also, since the area of the air supply inlet is smaller than the working section, the wind tunnel has a buffer space to which the stream, colliding with the building model, diverges (Figure 1). Because of the buffer space, the diverging stream does not reattach in the case of 0° (Figures 7 and 9). In a conventional wind tunnel, the diverging stream reattaches or, at least, flows near the wall surface and the flow, shown in Figures 6 to 9, does not appear. Thus, under the condition of a conventional wind tunnel, the turbulent kinetic energy tends to be overestimated in the standard k-ε model, because the turbulent kinetic energy is produced by strain. On the other hand, the diverging stream keeps straight in the BRI wind tunnel. It is thought that the diverging stream line maintains a lower turbulent kinetic energy in the calculation with the standard k-ε model.

Figure 10 shows the correlation between measurement and calculation of the wind pressure coefficient on each wall (i.e. 60 points * 4 walls) and the roof (144 points) (for 0° wind direction and no openings). It also shows the mean velocity and the rms of the turbulent velocity fluctuations for the coarse grid layout as shown in Figure 2 (555 points). The wind pressure coefficient shows strong correlation, although there is a little difference on the windward side (x-). The slope of the regression line is 0.81, and the calculated value is predominantly smaller than the measured value. However, the slope of the regression line is close to 1.0 at the mean velocity, although the correlation is weaker. The enclosed points in Figure 10b, which show the measurement value is smaller than the calculated value, are at the edge of the main stream (Figure 9 locations D).

In Figure 10b, points are shown in the range of the angle formed by the measured mean velocity vector and calculated vector. These two vectors corresponds relatively well at high velocity points. The rms of the turbulent velocity fluctuations shows weaker correlation than the mean velocity.

Results for a wind direction of 45° are shown in Figures 11 to 14. The calculated mean velocity field is relatively similar to the experimental field (Figure 12), although there are some differences in the mean velocity at the edge of main stream (Figure 12 locations E) and the experimental field has a little asymmetry (on the windward and leeward sides).

The calculated results show two low wind pressure areas on the roof extended from the windward corner as with the experimental results (Figure 11). These areas are formed by conical vortices, and conical vortices are shown in the calculation result (Figure 13) even though it is said that conical vortices do not usually appear in the standard k-ε model. The air flow field is thus satisfactorily calculated by using the standard k-ε model, and it is thought that the form of the wind tunnel is suitable for the calculation with the standard k-ε model as with the case of a 0° wind direction.

Figure 14 shows the correlations of wind pressure coefficient, mean velocity and the rms of turbulent velocity fluctuations at 45°. These results show a similar tendency to the 0° results. In particular, the wind pressure coefficient shows strong correlation, while the mean velocity and the rms of the turbulent velocity fluctuations have weaker correlations. The slope of the regression line of the wind pressure coefficient is under 1.0 and, generally, the calculation values are less than the measurement values.

Figures 15 (a) and (b) show the correlation coefficient and the slope of regression through the origin of the wind pressure coefficient \(C_p\) symbol in figure), the mean velocity \((U)\) and the rms of the turbulent velocity fluctuations \((SD)\) for the large (coarse) grid \((L)\) and the small (fine) grid \((s)\). The wind pressure coefficient has strong correlation for all the wind directions (0° to 45°), but the slope of regression is 0.8-0.9. This means that the calculated
a) Wind Pressure Coefficient  

b) Mean Velocity (Large Grid)  

c) $\sqrt{\text{Wind Pressure}}$ (Large Grid)  

Figure 10. Calculation vs. experimental measurement correlations ($0^\circ$, no opening).

a) Experiment  

b) Calculation  

Figure 11. Wind pressure coefficient ($45^\circ$, no opening).

a) Experiment [m/s]  

b) Calculation [m/s]  

Figure 12. Mean velocity ($45^\circ$, no opening. Z=1000 mm).
wind pressure coefficient is lower than the measured value. The slope of regression of the mean velocity is almost 1.0 (0.9-1.1) though the correlation coefficient is smaller than the wind pressure coefficient. Hence the calculated mean velocity has some difference from measurement but the general result can be regarded as good. Agreement between measured and calculated turbulent velocity fluctuations is not as good.

3.2 Air Flow Close to the Wall

Results were also used to assess the property of air flow close to the wall and the adequacy of the use of the Log-Law. Figure 16 shows the relation between the mean velocity component along the wall and the turbulent kinetic energy at points 50mm away from wall in the case of 0° wind direction. In the simulation, the closest points at which the Log-Law was applied were at 12.5mm away from wall. The following points were at 50mm away. The second closest points (i.e. those at 50mm from the wall) were used for comparison. By comparing measurement (Figure 16a) with calculation (Figure 16b) it can be seen that the results are relatively similar although the distribution along the side walls (y-, y+) are a little different and the calculated turbulent kinetic energy is a little higher on windward side (x-). The problem of the standard $k-\varepsilon$ model, in overestimating turbulent kinetic energy, $k$, in the collision area, appears a little.

The results suggest that when using the Log-Law as the boundary condition of the wall, it is possible to use the coarse mesh near wall in general. The curved line in Figure 16 shows the relation between the mean velocity and the turbulent kinetic energy that fills the Log-Law at a point 50 mm away from the wall. If the closest points are set 50mm away from the wall, the turbulent kinetic energy has the curve relation with the mean velocity in Figure 16, and the property of the local flow close to wall is different from the experimental air flow. This shows that the coarse mesh makes the local flow differ from the experimental air flow when the Log-Law is applied.

3.3 Ventilation Rate and Pressure

The pressure across an opening and the resultant ventilation rate have an important influence on the properties of indoor air flow. This study, therefore, compares the calculated pressure and ventilation rate with the experimental (measured) values. Figure 17 and 18 compares the measured and calculated ventilation rate for the model building with diagonally located openings (as illustrated in Figure 2). Ventilation rate is expressed for inflow at Opening A and outflow at Opening B. Both the measured and calculated ventilation rates were determined by integrating the velocity by area (where the measured velocity was based on measurements at 48 points and the calculated value was based on calculations at 162 grid points).

Figures 19 and 20 compare the calculated and measured differences in pressure coefficient between:

- Opening A and the floor;
- Opening B and the floor;
- Opening A and Opening B.

Where the wind pressure at the location of each opening was determined from the solid model (i.e. openings sealed) by averaging the pressure that was measured at the 6 pressure taps that represented that opening location (see Figure 4). The wind pressure on the floor was obtained by averaging the pressure that was measured at the 96 pressure points on the floor (see Figure 3).

As can be seen from Figures 17 and 18, the calculated ventilation rate showed generally good agreement with the measured value. At high ventilation rates the calculated value was approximately 10% above the measured value. The calculated differential pressure also showed a good relation to the measurement (Figure 19), but the calculated differential pressure is lower than the measured value (Figure 20). The simulation, tended to calculate the ventilation rate to be larger and the differential pressure to be smaller in value than the measurement.

The resultant variation in discharge coefficient at each opening is shown in Figure 21 where the discharge coefficient, $\alpha$, was determined from Equation 2 below:

$$\alpha = \frac{|Q|}{3600AU_0\sqrt{|\Delta C_p|}}$$

where $Q$ is the ventilation rate [m$^3$/h] in Figure 17, $A$ is the area of opening (=0.86*1.74[m$^2$]), $U_0$ is standard velocity (=3.0m/s), and $\Delta C_p$ is the difference of the wind pressure coefficient in Figure 19.
The indoor pressure was represented by the average pressure at floor level. By applying Equation 2 to the results it was found that when \( Q \) is large and \( \Delta C_p \) is small, the discharge coefficient, \( \alpha \), is high. This tendency is shown in Figure 21. The calculated composite discharge coefficient of Opening A and Opening B (\( \Delta C_p = C_{pa} - C_{pb} \)) shows a higher value than the measured value. Also the calculated discharge coefficient of a single opening shows significant error in some wind directions. It is thought that this is because the \( C_p \) floor value (\( C_{pf} \)) is calculated under no openings condition (i.e. the solid model) and \( C_p \) at the opening (\( C_{pa}, C_{pb} \)) was calculated under with-openings condition, that is, the difference of \( C_p, \Delta C_p \) under other conditions results in a significant error of discharge coefficient in Equation 2.

### 3.4 Air Flow in the Building Model

The results of calculated and measured air flow within the scale building model are summarized in Figures 22 to 27. In the case of 0° wind direction, the calculated mean velocity field generally corresponds well with the measured flow field. The main areas of discrepancy are the inflow at Opening A (Figure 22 location F) and in the main stream after collision with the partition wall (Figure 22 location G).
The mean velocity at 0° has strong correlation with the indoor grid (980 points) (Figure 23). However, the rms of the turbulent velocity fluctuations shows weaker correlation than the mean velocity (Figures 24 and 25).

In the case of a 105° wind direction, the calculated ventilation rate (670m³/h) is extremely low compared to experimental measurement (1,680m³/h) (Figure 17). Also the calculated mean velocity is smaller at 30% less than measurement (Figure 27). This is because the calculated differential pressure at Opening B, $\Delta C_{pB,Calc}$, (inflow side, Figure 26 location H) is almost zero as opposed to the experimental value($\Delta C_{pB,Exp}$=0.15) (Figure 19). The reason is that the calculated distance to which the separated flow reattaches is shorter than the measured value (reattachment point: Figure 26 J) and the calculated air flow is parallel to Opening B (similar to an air curtain). Uncertainty of the separated flow, (which is calculated by the standard $k$-$\varepsilon$ model), appears in this case. This shows that air flow around the building must be properly calculated to simulate a good indoor air flow field.

Figure 28 shows the correlation coefficient and the slope of regression through the origin of the wind pressure coefficient ($C_p$), the mean velocity (U) and the rms of the turbulent velocity fluctuations (SD) for the coarse grid (L), and the fine grid (S: no-opening, so: with opening) and the indoor grid (I). The mean velocity around the model shows strong agreement and the slope of regression is almost 1.0. However the mean velocity inside the model has weaker correlation and the slope is less than 1.0 in some cases (i.e. for U-I: at wind directions of 75°, 105°, etc.).
The calculated turbulent velocity fluctuation has weaker correlation than the mean velocity. In particular there are large differences inside the model (especially SD-I). The slope of regression of the turbulent velocity fluctuations has the tendency to be greater than 1.0 (the maximum is about 1.4) close to the wall on the fine grid (SD-s, -so). This result identifies the major problems of the standard \( k-\varepsilon \) model as the overestimation of the turbulent kinetic energy, but the overestimation is restricted, and the air flow field is not influenced too much in this case. The slope of the indoor turbulent velocity fluctuations (SD-I) is under 1.0 thus the calculated indoor turbulence has a smaller value than the measurement values.

4. Conclusion

In this study, the air flow field in the BRI wind tunnel was calculated by CFD analysis, using a standard \( k-\varepsilon \) model, to compare with the experimental air flow field both within and around a scale building. The adequacy of the calculation was examined and the conclusions are as follows:

- The calculated air flow field was generally in good agreement with measured field;

- The distribution of the wind pressure coefficient is similar, on the whole, between calculation and measurement, but the calculated wind pressure coefficient tended to be slightly lower than measurement (at approximately 80-90% of the measured value);

- The mean calculated velocity around the building model was relatively similar to the measured velocity although there were some differences in the wake region as well as at the edge of main stream and at the reattachment points;

- The calculated turbulent kinetic energy around the model had weaker correlation than the mean velocity. Also there was slight overestimation of the turbulent kinetic energy on the windward side (this is a main issue of the standard \( k-\varepsilon \) model). However this did not influence the simulated air flow field too much. It was thought that the properties of the wind tunnel including the buffer space and honeycomb baffle assisted in providing conditions that gave good agreement.

- The calculated property of local flow close to the wall was relatively similar to measurement, but it is necessary to note that a coarse mesh may result in a difference of local flow from the experiment;

- The calculated ventilation rate and differential pressure between openings were in good agreement with experimental measurement. However, the calculated ventilation rate was greater than the measured value although the calculated differential pressure was found to be lower than the measured value;

- Indoor mean velocity and turbulent kinetic energy were found to only have weak correlation compared with outside. This exemplified that the calculated indoor air flow was strongly influenced by the external conditions;

- In the case of the BRI wind tunnel, the air flow field, calculated by CFD analysis, using the standard \( k-\varepsilon \) model, showed moderately good agreement with measured result. However, the \( k-\varepsilon \) model is generally said to have significant difficulties, and it is important to check the calculated results when using the \( k-\varepsilon \) model.

References

