UDC: 628.8: 533.6.07 Keywords : Comparison study,
Computational fluid dynamics, Wind tunnel.

Investigation of Natural Ventilation with Computational Fluid Dynamics A Comparison Study with Wind Tunnel Results

J. Kindangen*° and G. Krauss*

This paper presents an investigation into natural ventilation in the field of computational fluid dynamics using in particular rather rough mesh cells. The CFD results were then compared to the wind tunnel results obtained by Gouin at the Centre Scientifique et Technique du Bâtiment (CSTB) in Nantes. The role of eaves, and that of window configuration on windward and leeward sides of buildings was also investigated to search for a better interior airflow.

1. Introduction

Increasing demands for energy saving, and a higher degree of comfort in rooms compel designers to use more sophisticated analysis methods, computational fluid dynamics (CFD). Numerical simulation of turbulent fluid flow in rooms and theirs environments is a modern design tool enabling designers to analyse fairly rapidly the distribution of fluid flows, as compared to the time needed for full or reduced scale experiments such as wind tunnel investigation.

Cross-ventilation usually means natural ventilation with a rather large inflow caused by wind through open windows or doors. The wind blows directly into the indoor space with small loss of dynamic energy. Cross-ventilation offsets internal and solar heat gain : occupants are directly cooled down by increasing convective and evaporative heat loss from their body surface, and the temperature of building structure is reduced at night [1]. The cooling of occupants depends upon the distribution of indoor airflow, greatly influenced by the wind circulating around the building.

This paper presents the results of the numerical simulation and compares them to the results of wind tunnel investigation by Gouin [2].

* Centre de Thermique de l'INSA de Lyon/Equipe Equipement de l'Habitat, Bâtiment 307, 20, Avenue Albert Einstein, 69621 Villeurbanne Cedex, France.
°Faculty of Engineering, Department of Architecture, University of Sam Ratulangi, Jl.Kampus Unsrat Bahu, Manado 95115, Indonesia.

2. Experimental Methods

2.1. Simulation of the Natural Wind

In this study, we have used the flow analysis package STAR-CD, and the standard k- ε turbulence model is implemented to account for turbulent flow. The profile of natural wind was derived from Gouin's study, which used a boundary layer wind tunnel at CSTB in Nantes. The turbulence characteristics and vertical profile of wind speed are influenced by atmospheric stability and by the type of terrain, which is a function of terrain roughness. The vertical profile of wind speed was given by a power law according to the following formula :

$$\frac{V_z}{V_{zi}} = \left(\frac{z}{z_i}\right)^{\alpha}$$

where :

Vz = mean wind speed at height z (m/s)

Vzi = mean wind speed at reference height zi (m/s)

z = height above ground level (m)

zi = reference height (m)

α = an exponent characteristic of terrain roughness

Gouin's study led us to choose α value of equal to 0.16. The flow and boundary conditions are isothermal. The reference wind velocity is 5.843 m/s at a reference height of 4.25 m with a reference turbulent intensity of 22%. Figure 1 presents the profile of vertical of wind velocity applied at inlet. Our simulations were done with very rough mesh cells. A coarse grid is sufficient to provide a good qualitative prediction of the flow fluid [3,5].

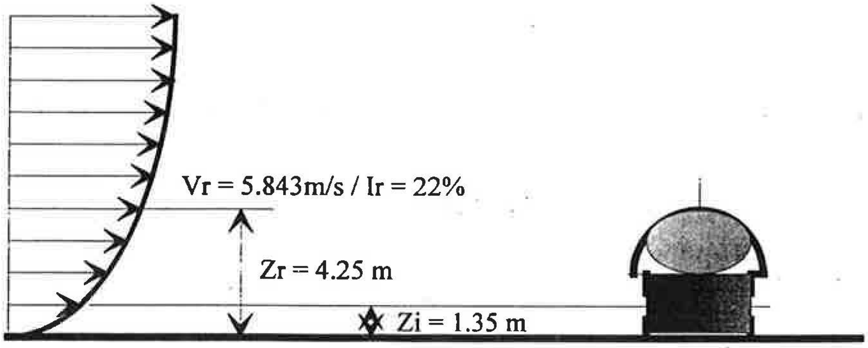


Figure 1. Vertical profile of wind speed at inlet

The dimensions of the cube where the building model was placed (as shown in appendix) are : length (x) : 32.00 m, width (z) : 32.00 m, height (y) : 7.50 m. It was subdivided into x-, y- and z-directions, the number of volume elements increasing from 2244 to 7200 cells, or with mesh sizes as shown in the Table below.

Table 1. Variation of cell dimensions (in m.)

x-direction		y-direction		z-direction	
min.	max.	min.	max.	min.	max.
0.44	3.50	0.30	1.50	0.30	1.50

Wall, roof and ceiling of buildings were represented by baffles, considered as impermeable. The CFD results were taken when convergence had been reached. The results of three spatial components of velocity were used, with slices of 1.35 m height.

2.2. Building Models

The simulation of most models followed Gouin's study , with the exception of models 3, 4, and 9. For this purpose all tested models were set up with the following dimensions : length : 7.00 m, width : 7.00 m, height of ceiling : 2.70 m. Windows sizes varied from a wall porosity of 12.78% to 40% for windward and leeward sides (Figure 2). Wind angles

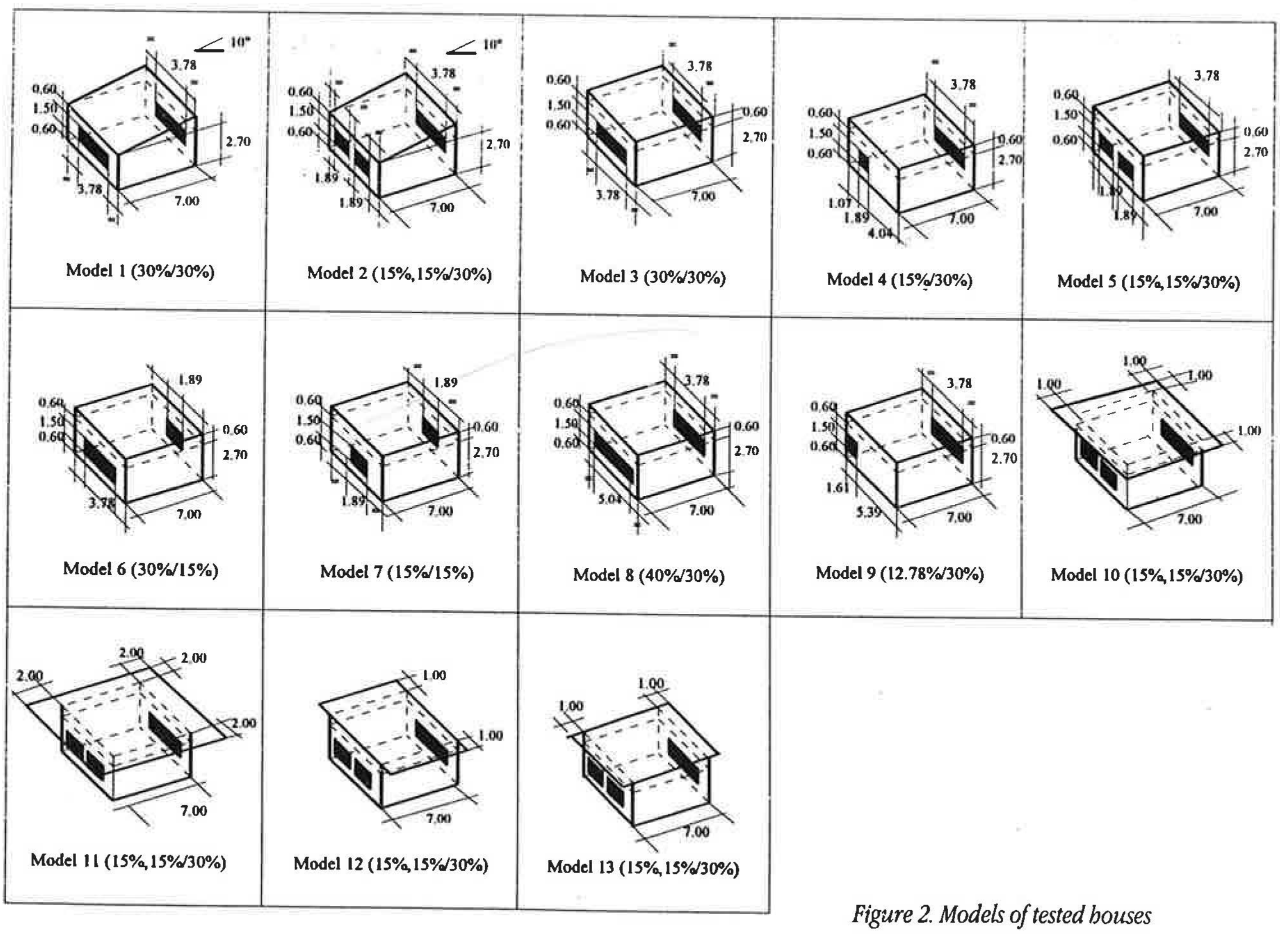


Figure 2. Models of tested houses

have been also varied at 0°, 30°, 45°, 60°, 90°, 120°, 135°, 150° and 180°. Each model was placed in the centre of the floor of a cube, inlet and outlet being applied on the sides. The ground was simulated as a impermeable wall.

2.3. Results Processing

For each model tested, the following two non-dimensional indoor air motion parameters were computed, based on some points measured according to the distribution of grid :

$$C_v = \frac{1}{n}\sum_{i=1}^{n} \frac{V_{i(x,y,z)}}{V_r}$$

$$C_{v_{max}} = \frac{V_{i(x,y,z)_{max}}}{V_r}$$

where :

Cv	=	average velocity coefficient
Cv max	=	maximum local average velocity coefficient
Vi(x,y,z)	=	mean velocity at interior location i in three spatial components (m/s)
Vr	=	mean outdoor reference free-stream velocity at the height of 4.25 m (m/s)
n	=	number of points measured in the model

Cv is the measure of the relative strength of the interior air movement in the horizontal plane, which is representative of the occupied space of the room, in this case 1.35 m above the floor. We also observed, for each model, the maximum Cv, as it plays an important role when analyzing jet and blockage effects.

3. Results and Discussion

Comparison of Results

The figure below compares the average velocity coefficients resulting from wind tunnel and CFD. The correspondence between the numerical simulation and the experimental results is in the whole fairly close, although, for a wind incidence of 90°, the figures are surprising : the relative difference being 49.36%. For wind angles of 45°, 60° and 150°, the results are well matched (relative difference of less than 10%). For all results, the relative difference is 25.34%. The discrepancy obtained is apparently due to the real symmetry achieved by CFD, while it is extremely difficult to place the object in absolute symmetry in a wind tunnel. This was proved by a 90° wind angle simulation with an approximate inclination of 5°: the match was then deemed to be good, less than 10% the relative difference. This explains why it was not considered necessary to investigate every efect of the 90° wind angle. In addition, in the other cases the discrepancies were due to differences in the location of the points measured, these being difficult to simulate in CFD with a distribution close to that of wind tunnel experimentation.

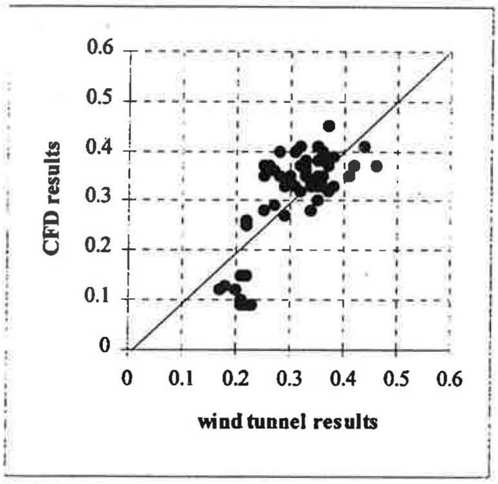
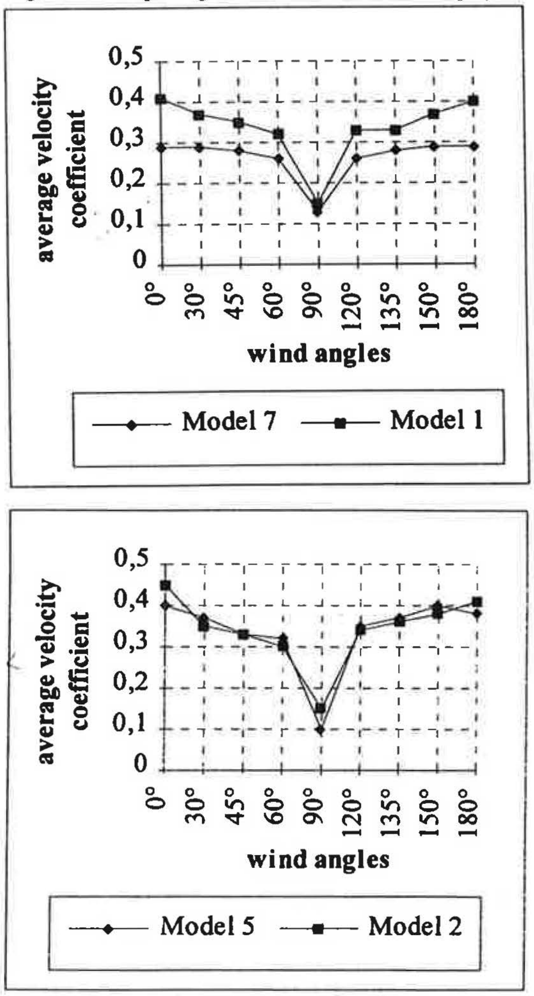


Figure 3. Comparison of wind tunnel and CFD results Cv results. $R^2 = 0.802$ without wind incidence of 90° results

Exploitation of Results

Even though the simulated models were restrictive, they were sufficiently representative to enable comparison of some parameters, e.g. roof geometry and configuration of opening. The influence of roof geometry was investigated as follow. To compare the different roofs, we have shown the increase of Cv, in percentage, between the flat roof type and the single-sloped roof type respectively keeping the same wall porosity and wind angles.

Figure 4(a) presents better results given by single-sloped roof over flat roof, the average velocity coefficients for sloped roofs are always higher than those for flat roofs; on the contrary, Figure 4(b) shows that flat roofs give better results for most wind angles, with the exception of wind angles of 0° and 180°, where the Cvs of sloped roof are slightly higher. Therefore, the configuration of opening on the windward side also plays an impor-



(a)

(b)

Figure 4. (a) (b) Comparison of flat roof and single-sloped roof Cv results

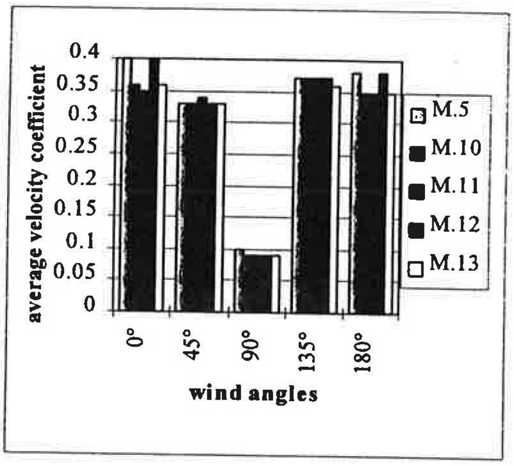


Figure 5. Effects of the eaves of buildings

tant role on interior airflow improving, even with the same wall porosity. In other words, the design variables are not independent of each other.

Figure 5, we can see that the presence of eaves decreases slightly the average indoor velocity coefficient, by about 10%, in particular for wind angles of 0° and 180° (comparison of model 5 and 10); in the case of models 10 and 11, the dimensions of eaves have no significant influence, while comparison of models 12 and 13 shows that the presence of eaves on the windward and leeward sides has a more significant influence than the presence of eaves on the other sides.

Fixing a constant opening on leeward side enabled us to analyze the influence of window configurations on the windward side. The model 3 was taken as a reference model. Comparison of models 3 and 5 in the Figure 6(a) shows that the division of the opening into two windows (the same wall porosity on facade) increases the Cv, by about 6.40%, inversely, the division of the opening causes a lower maximum Cv than in the case of undivided opening . Naturally, the change in window size on the windward side causes a modification which increases as the opening becomes larger. In this case, with an opening smaller on the windward than on the leeward side, we observe that the maximum Cv increases, this is due to the jet effect. If the opening is not in the axis of symmetry of the building, the Cv increases while the maximum Cv decreases.

Conversely, when fixing the windward opening, the influence of window configurations on leeward side could be analyzed. Figure 6(b) shows that the division of the opening on leeward side has less influence on Cv and maximum Cv. Effects of blockage are produced, since the opening on leeward is smaller than on windward. There is also the jet effect if the opening on leeward is bigger than on windward. By comparing model 6 (Figure 6a) to model 8 (Figure 6b) it can be seen that, if the opening on the windward and leeward sides are simultaneously enlarged, Cv increases by about 25% and maximum Cv decreases by about 1.20%. In other words, in model 6 (Figure 6a), the jet effect is more significant than in model 8 (Figure 6b).

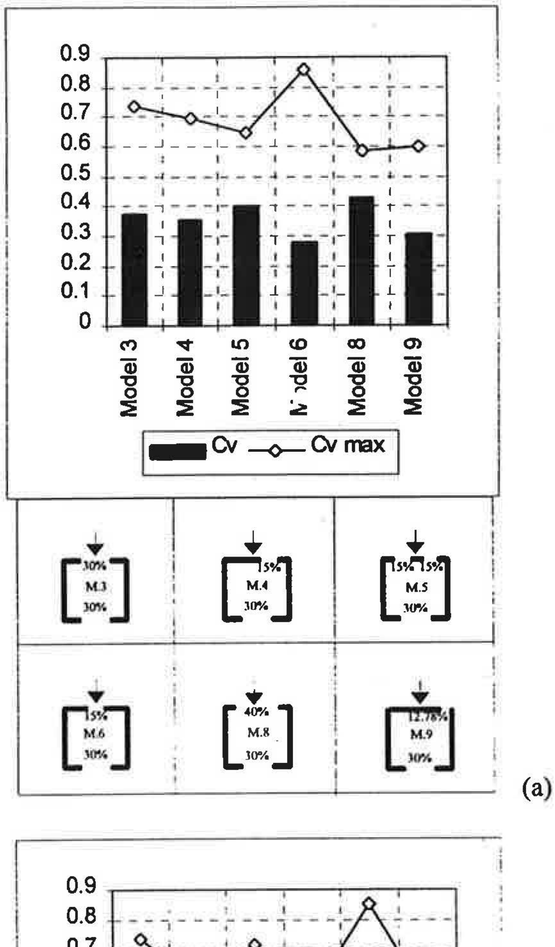


Figure 6.(a) (b) Influence of wall porosity and of the positions of opening

4. Conclusions

Average velocity coefficients for wind tunnel and CFD results were compared for some the models. The main findings of this study are as follows:

In computational fluid dynamics the processing of a 90° wind angle should take into account the effects of symmetry, this could be considered as a necessary precaution when comparing with results of wind tunnel experimentation. Moreover, it is suggested that the points measured for these calculations have to be perfectly matched. Thus it was confirmed that numerical simulation of the 3-D turbulence in a building ventilated by means of a k-ε model corresponds closely to the experimental results.

Numerical prediction of room airflow has many advantages, such as time saving and has been shown to be a very promising technique, it is to be noted that it is not generally easily applied at the design stage because CFD codes are complicated to use and require some considerable experience if one is to expert results which can be trusted. Thus, more research is needed to find a more serviceable design tool in this field, i.e. expert systems.

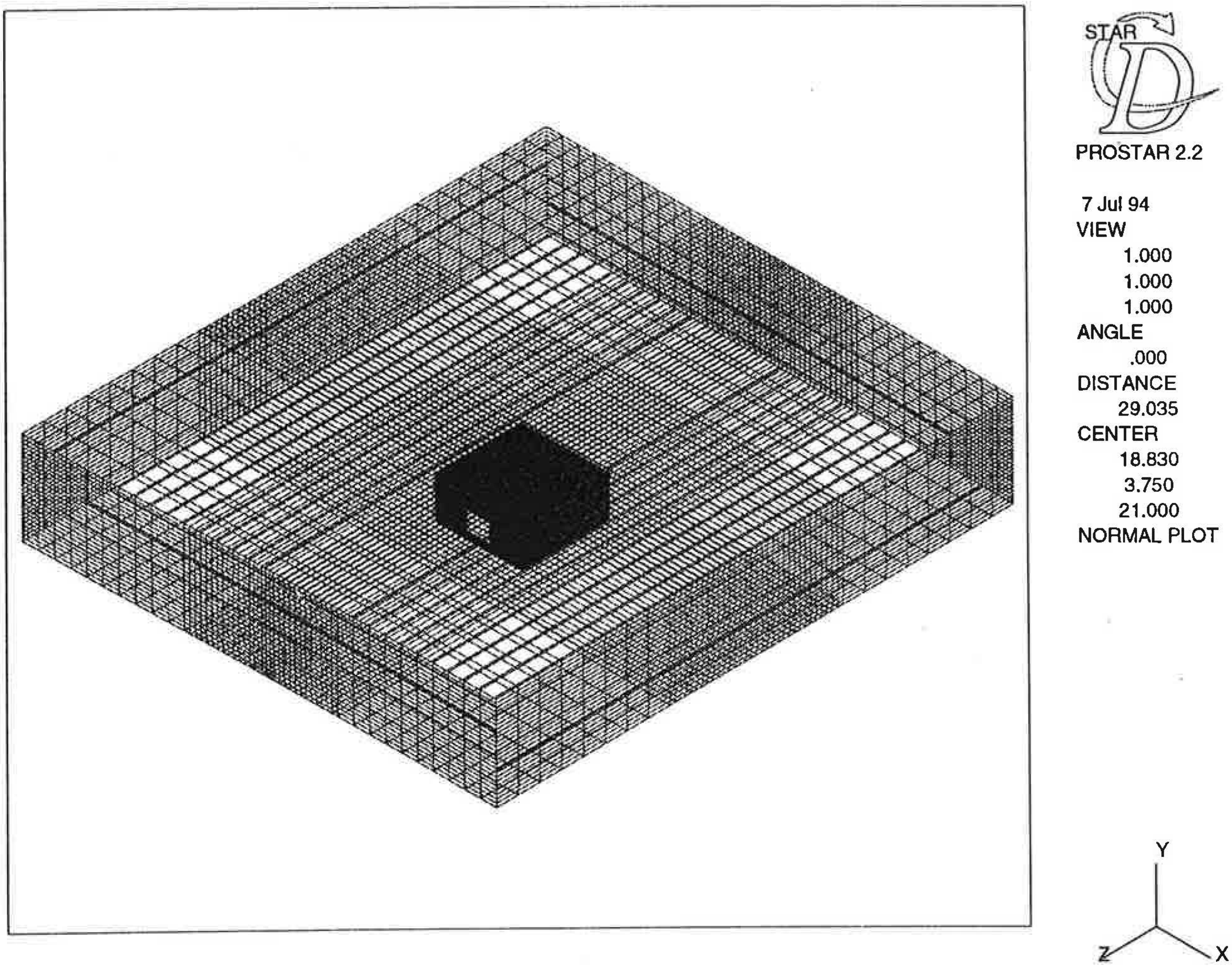
5. References

1. Ernest, D. R., Bauman, F.S., Arens, E.A., 1991, "The Prediction of Indoor Air Motion for Occupant Cooling in Naturally Ventilated Building", *ASHRAE Transactions*, Vol. 97 Part 1, pp. 539-552.
2. Gouin, G., 1984 "Contribution Aérodynamique à l'Etude de la Ventilation Naturelle de l'Habitat en Climat Tropical Humide", Thèse de Docteur, Université de Nantes.
3. Kindangen, J.I., 1994, "Modélisation Aéraulique en Climat Tropical Humide, d'un Bâtiment Largement Ouvert sur l'Extérieur". Mémoire de DEA, CETHIL/EEH, Institut National des Sciences Appliqués de Lyon.
4. Celestine, C., 1985, "Ventilation Naturelle et Confort Thermique dans l'Habitat en Climat Tropical Humide - Approche Expérimentale en Vraie Grandeur de la Ventilation Naturelle sur un Site de la Guadeloupe - Etude Comparative en Soufflerie", Thèse de Docteur-Ingénieur, INSA de Lyon.
5. Borth, J., Suter, P., 1994,"Influence of Mesh Refinement on the Numerical Prediction of Turbulent Air Flow in Rooms", *Proceedings of Roomvent 1994, Air Distribution in Room, Fourth International Conference*, Craców, Poland. Vol. 1 pp. 138-148.
6. Tsutsumi, J., Katayama, T., Hayashi, T., He, P., 1992, "Numerical Simulation of Cross-Ventilation in a Single-Unit House", *International Symposium on Room Air Convection and Ventilation Effectiveness, Tokyo*, pp. 447-451.
7. Murakami, S., Kato, S., 1989, "Numerical and Experimental Study on Room Airflow 3D Prediction Using k-ε Turbulence Model", *Building and Environment*, Vol. 24 no. 1, pp. 85-97.
8. STAR-CD, 1993, Computational Dynamics Limited, London.
9. Jones, P.J., Whittle, G.E., 1992, "Computational Fluid Dynamics for Building Air Flow Prediction - Current Status and Capabilities", *Building and Environment*, Vol. 27 no. 3, pp. 321-338.

Appendix

The illustration of STAR-CD results:

(a) Geometry of Grid.



(b) Vectors of velocity magnitude for wind incidence of 0°