Enhancement of cross-ventilation of a detached house using roof surfaces in densely populated urban areas Part 2 Numerical investigations about the effects of the roof surface use by CFD

Y. Kadowaki, T. Kurabuchi, T. Nonaka
Tokyo University of Science, Japan

T. Endo
Kanto Gakuin University, Japan

ABSTRACT

In Part 1, the changes in wind pressure coefficient distribution on the surfaces of the buildings when building coverage is varied are evaluated and identified, and it is elucidated that the ventilation through skylight is effective because the surface pressure can be kept negative on roof surfaces even when building coverage is high.
In Part 2, the results of numerical simulation (CFD) are compared with the experimental results. To perform this study, for the purpose of analyzing the configurations such as the roof with gradient, mesh building methods based on multi-blocks and layer meshes are proposed to generate calculation meshes in optimal and systematic manner, and the results of the methods are verified. Further, because the validity of the modeling of urban area in cyclic boundary condition is already reported in previous studies, the case where building coverage of urban area is changed is analyzed by varying the size of the area under computation and by the method to apply cyclic boundary condition, and the results are compared with the results of experiment as described in Part 1 of the study.

1. INTRODUCTION

To achieve good cross-ventilation performance, it is indispensable to perform the study based on highly accurate wind pressure coefficient distribution. In the previous study, the accuracy to predict wind pressure coefficient distribution was verified by analyzing the airflows around buildings with complicated shapes by using hybrid meshes, which consist of tetra-meshes and hexa-meshes (Endo, 2005). In this case, some of the problems remained unsolved. The problems are: even when turbulent flow model may be improved, the prediction accuracy is poor on the roof with gradient, and it is necessary to develop a mesh system, which can cope with the evaluation of complicated shapes and which provides high prediction accuracy. To solve the problems, we propose a mesh system, which uses both multi-block method and layer mesh method. In the multi-block method, the area to be analyzed is divided to several blocks, and each block is subdivided to hexa-meshes. Because this shows good boundary adaptability, this method can be conveniently used in the extension to urban area. The layer mesh method is to build meshes in form of layers around the shapes to be analyzed, and it is characterized in that the first meshes located next to wall can be easily adjusted. In Chapter 2 of the study, an evaluation is made on a method, by which highly accurate analysis can be conducted without depending on the resolution of meshes. In Chapter 3, the validity of the method is verified by comparing the results of the study with experimental results on the detached house model. Further, in Chapter 4, based on previous study (Endo, 2007), which demonstrated the validity of the modeling of urban area in cyclic boundary condition, simulation is performed on the experiment of Part 1, and by comparing the results, assessment is made as to whether it is also effective or not for the analysis of urban area using cyclic boundary condition.

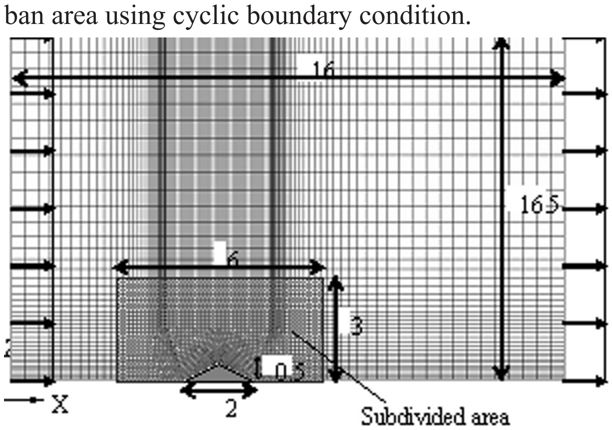

Figure 1: Flow Geometry

Table 1: General Features of Analysis

Algorithm		SIMPLE
Differencing scheme		QUICK
Boundary conditions	Inlet	Uniform
	Outlet	The gradients of all variables are zero
	Top and Sides of domain	The normal component of velocity and the normal gradient of all other variables are zero
	Bottom of domain	Free slip

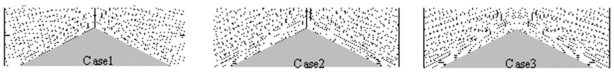

Figure 2: Mesh building case

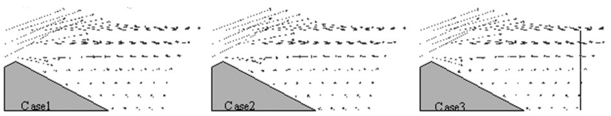

Figure 3: Wind velocity vector distributions (15 times standard cell number)

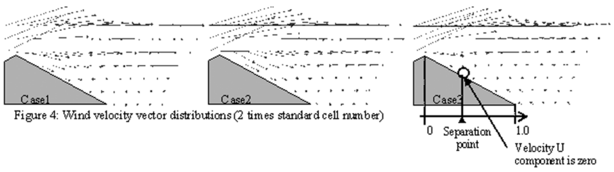

Figure 4: Wind velocity vector distributions (2 times standard cell number)

2. EXAMINATION OF TWO-DIMENSIONAL ESHES

2.1 Outline of CFD

When simulation study is performed on the airflows around buildings with complicated shapes as seen in general type housing, problem arises in the method to build meshes on and around roof surfaces with gradient. In this chapter, as a basic study to evaluate airflows on two-dimensional topography, which is bent like a roof as shown in Figure 1, an evaluation is made by using multi-block method on the mesh building method, which is hardly influenced by the evaluation of meshes. The conditions of computation are given in Table 1. The mesh building method is divided to three cases (Fig.2). In Case 1, only the multi-block method is used. In Case 2 and Case 3, the multi-block method is simultaneously used with a method to build layer meshes to accurately follow the shape of the object to be evaluated. To divide the layer mesh portion, it is divided in such manner that the portion near the roof ridge is divided vertically in Case 2. In Case 3, it is divided crosswise to exactly following the shape of the roof surface under study. In each of the cases, the portion under study is further subdivided so that the number of cells in the hatched area shown in the Figure 1 will be 2 times, 4 times and 15 times respectively of the standard division shown in Figure 2. By setting pseudo-turbulent flow with Reynolds numbers to be at 100 and the boundary condition of roof surface to be slip, meshes are subdivided regardless of the value of y+.

2.2 Results and Discussion

Figure 3 shows wind velocity vector distribution in each case when the number of cells in each portion of the area under study is subdivided so that the total number of cells will be about 360k, i.e. 15 times as many as that of the standard mesh, in which total number of cells is about 5k. In this case, there is almost no difference between the cases, and this is defined as an exact solution of the field of flow. Figure 4 shows wind velocity vector distribution in each of the cases as subdivision is made when the number of cells on the portion of the area under study is increased to two times of the standard mesh (i.e. total number of cells: about 10k). In Case 1 and Case 2, airflows along wall surfaces are still extensive even after passing through the top of the surface and recirculating wake is not generated almost at all. On the other hand, in Case 3, the separation of airflows occurs near the top and recirculating wake appears as shown in Figure 3, and this may be a case, which is the closest to the solution. Figure 5 represents the positions of separation points in horizontal direction by taking total number of cells on the axis of abscissa and the position of apex on the axis of ordinate. When the meshes are at the finest, the positions of the separation points agree with each other in all cases. Even when the meshes are not fine, the positions of the separation points are almost the same in Case 3. Therefore, when layer meshes are built crosswise along the gradient, it would be possible to perform analysis with high accuracy even when relatively coarse meshes may be used. This may be attributed to the fact that meshes are arranged along the airflows on the crooked portion.

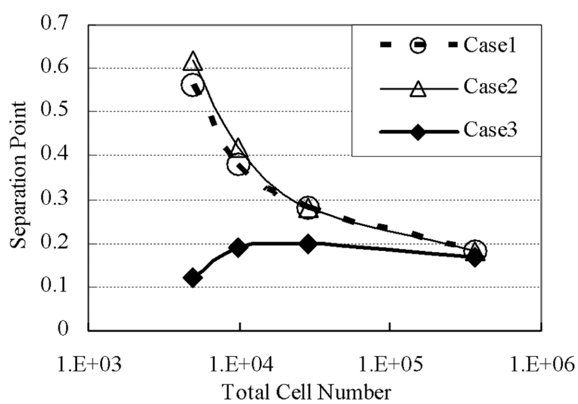

Figure 5: Separation point

3. EXAMINATION OF THREE DIMENSIONAL MESHES AROUND THE DETACHED HOUSE

3.1 Outline of CFD

Based on the results described in Chapter 2, analysis on three-dimensional turbulent flow is performed on a model of a detached house, and the results are compared with the experimental results. Figure 6 shows the flow area and the building under study. The conditions of computation are given in Table 2. The area to be analyzed is divided in multi-blocks. The method to divide to layer meshes on and around wall surfaces of the building is evaluated in Case 2 and Case 3 to correspond to the description in Chapter 2 as shown in Figure 7 As the turbulent flow model, an improved k-ε model, incorporating Durbin limiter (α = 0.8) (Durbin, 1996), is used by following the previous study.

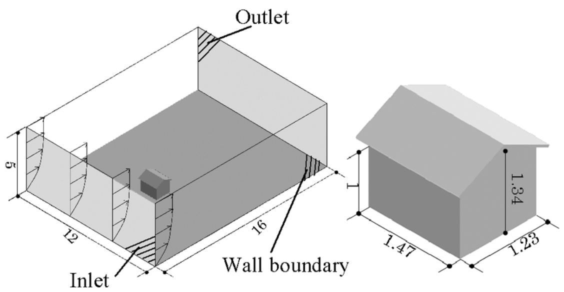

Figure 6: Building shape and flow geometry

Figure 6: Building shape and flow geometry

Table 2: General Features of Analysis

Algorithm		SIMPLE	
Differencing scheme	u, v, w	QUICK	
	k, ε	MARS	
Boundary conditions	Inlet	Wind tunnel experiment	
	Outlet	The gradients of all variables are zero	
	Top and Sides of domain	The normal component of velocity and the normal gradient of all other variables are zero	
	Bottom of domain	Wall boundary	Wall function

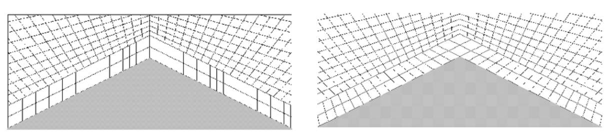

Figure 7: Mesh around the ridge

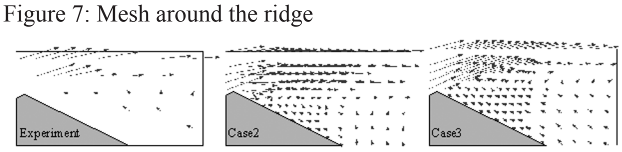

Figure 8: Wind velocity vector distributions near the roof ridge on wake region

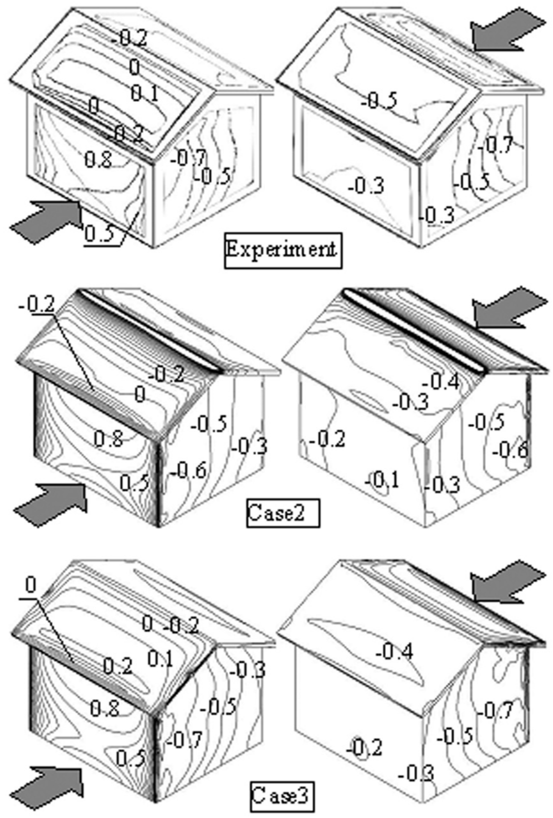

Figure 9: Wind pressure distribution

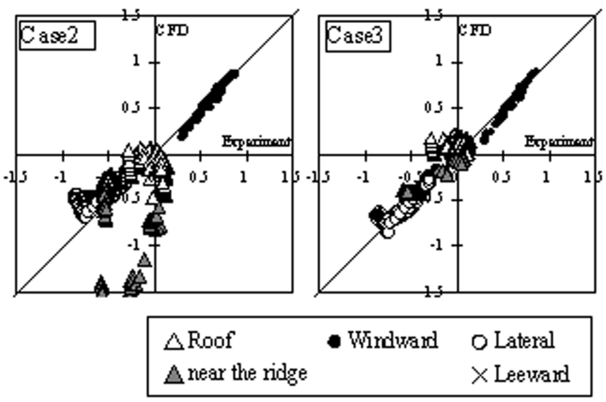

Figure 10: Correspondence of observed and predicted wind pressure coefficients

3.2 Results and Discussion

Wind velocity vector distribution near the roof ridge on wake region obtained by experiment and computation are shown in Figure 8. According to the previous study, it is difficult to reproduce the separation of airflows by CFD, and sufficient accuracy to predict wind pressure coefficient cannot be obtained. In Case 2, the airflows after passing the roof ridge run as if the flows adhere to roof surface, and then, the airflows are separated. On the other hand, in Case 3, the airflows are separated at the roof ridge in the same manner as in the results of the experiment and the direction of airflows also corresponds to the direction in the experiment. With regard to the turbulent flow energy, it is also confirmed that Case 3 shows the trend closer to that of the experiment such as the peak value. Figure 9 represents wind pressure coefficient distribution. On the surfaces except roof surfaces, there is almost no difference between Case 2 and Case 3, and these results generally match well with the results of the experiment. However, the results on the roof surfaces are generally lower in Case 2 compared with the experimental results. In particular, extreme negative pressure is generated near roof ridge. In contrast, in Case 3, the extreme negative pressure near the roof ridge is eliminated, and the accuracy to predict wind pressure coefficient distribution on roof surfaces is significantly improved as shown in Figure 10.

4. ANALYSIS OF AIRFLOWS AROUND THE UNIFORMLY SETTLED URBAN AREA WHEN BUILDING COVERAGE IS CHANGED

4.1 Outline of CFD

The airflows on and around the uniformly settled urban area are analyzed according to the result of the evaluation performed so far. As previously reported (Endo, 2007), the uniformly settled urban area can be reproduced well by using cyclic boundary condition. Thus, cyclic boundary condition is applied on inflow surface and outflow surface. The objects to be analyzed are: no gradient roof, 27°gradient roof shown in Part 1 of the study, and computation is performed with building coverage of 10%, 20% and 40% respectively. On the model with 27°gradient roof, the area to be analyzed is divided to multi-blocks as described in the preceding chapter. For the portion with layer meshes around wall surface of the building, the division method of Case 3 is used, which showed better agreement with the experimental results as described in the preceding chapter. Because wind direction is 0°, computation is performed only on symmetrical half-area. The building coverage is reproduced by changing the size of the area under computation as shown in Figure 11. As the turbulent flow model, the improved k-ε model, incorporating Durbin limiter (α = 0.8). The conditions of computation are given in Table 3.

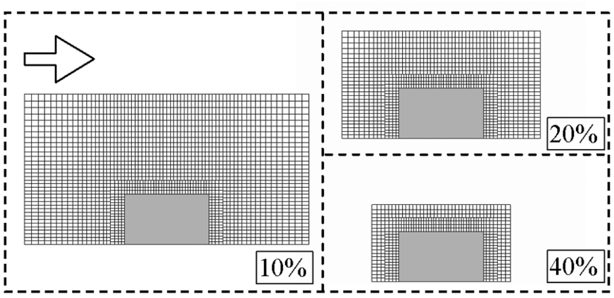

Figure 11: Flow geometry (horizontal plane)

Table 3: General Features of Analysis

Algorithm		SIMPLE	
Differencing scheme	u, v, w	QUICK	
	k, ε	MARS	
Boundary conditions	Inlet and Outlet	Cyclic boundary conditions	
	Top and Sides of domain	The normal component of velocity and the normal gradient of all other variables are zero	
	Bottom of domain	Wall boundary	Wall function

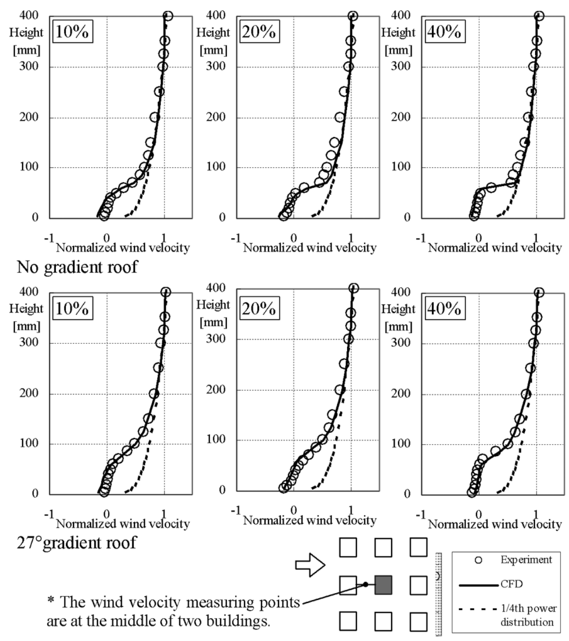

Figure 12: Wind velocity distribution in vertical direction

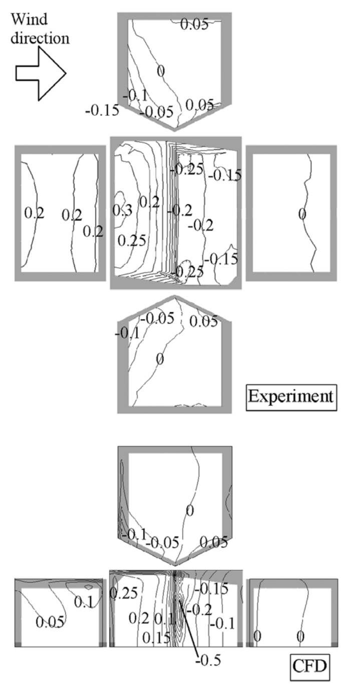

Figure 13: Wind pressure distribution (27°gradient roof – 20%)

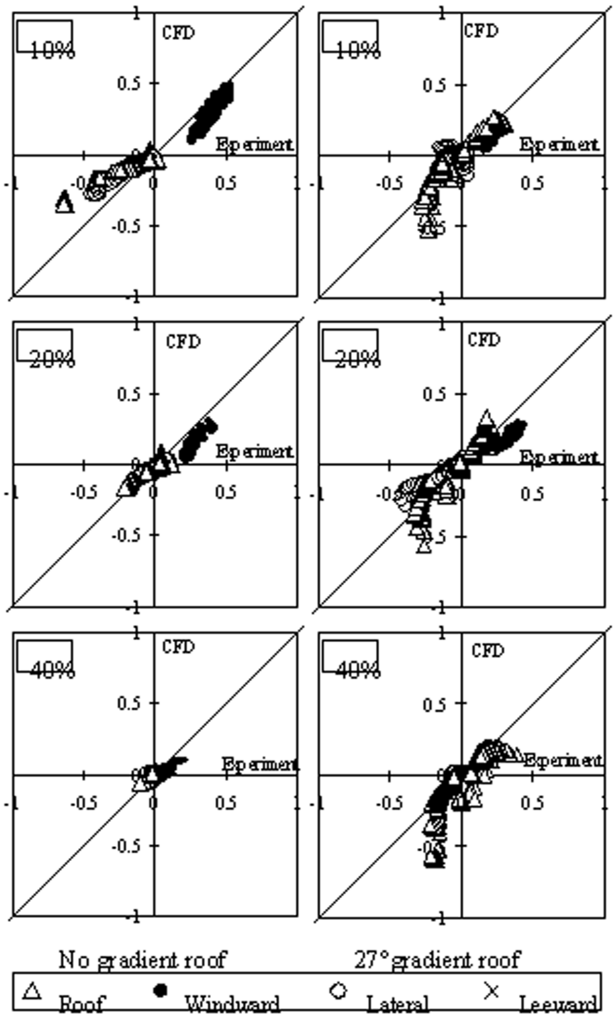

Figure 14: Correspondence of observed and predicted wind pressure coefficients

4.2 Results and Discussion

Figure 12 shows wind velocity distribution in vertical direction at the positions of wind velocity measuring points. In each case, the results show good matching with the experimental results. In upper space region, the results cope well with 1/4th power distribution. An example of wind pressure coefficient distribution is shown in Figure 13. Compared with the results of the experiment, there is a tendency that the prediction of wind pressure is generally low. Figure 14 represents the comparison of wind pressure coefficient between experiment and computation. When building coverage is decreased, the prediction of negative pressure is lowered in case of no gradient roof. In case of 27°gradient roof, negative pressure prediction is higher near the roof ridge on wake region side even when meshes are improved, and this indicates that further study would be needed. However, it may be admitted that the results show good matching with the experiment in other points.

5. CONCLUSIONS

The results obtained from this study are summarized as follows.

− By a new mesh-building method applying both multi-blocks and layer meshes, it was elucidated that the accuracy of the analysis on airflows on and around roof with gradient can be improved.

− By building the meshes crosswise along the roof gradient, it was found that the experimental results could be accurately reproduced even when relatively coarse meshes were used.

− By applying cyclic boundary condition and the new layer mesh building method, wind pressure coefficient on and around building was predicted in the uniformly settled urban area, which consists of building models closer to the reality, and the accuracy of this method was confirmed.

− Even when building coverage was changed, it was demonstrated that the application of cyclic boundary condition was satisfactory and valid.

REFERENCES

Durbin, P.A., (1996). On the k-ε stagnation point anomaly2, Int. J. Heat and Fluid Flow, 17, (1)

Endo, T., Kurabuchi, T., Ishi, M., Komamura, K., Maruta E., and Sawachi, T., (2005). Study on the Numerical Predictive Accuracy of Wind Pressure Distribution and Air Flow Characteristics - Part1 Optimization of Turbulence Models for Practical Use; Part2 Prediction Accuracy of Wind Pressure Distribution of Various Shaped Buildings. The International Journal of Ventilation Volume4 Number3: pp. 269-284

Endo, T., Kurabuchi, T., Nonaka, T., Kadowaki, Y., and Ohba, M., (2007). Numerical analysis of wind pressure distribution on a building in densely settled urban area by using CFD. 10th International conference of air distribution in rooms ROOMVENT2007: (in printing)

Murakami, S., Hibi, K., and Mochida, A., (1990). Three dimensional analysis of turbulent flowfield around street blocks by means of large eddy simulation part1. Journal of Archit. Plan. Environ. Engng. AIJ, No.412: pp. 1-10.