NUMERICAL SIMULATION OF TURBULENT FLOW AROUND A BUILDING COMPLEX

Hak-Sun Kim[*], Sungsu Lee†, Choon-Bum Choi†† and Kyung-Soo Yang†

[*]Department of Structural Systems and Computer-aided Engineering
Chungbuk National University, Cheongju, 361-763, Chungbuk, South Korea
e-mails: haksun@cbnu.ac.kr

† School of Civil Engineering
Chungbuk National University, Cheongju, 361-763, Chungbuk, South Korea
e-mail: joshua@cbnu.ac.kr,

†† Depaprtment of Mechanical Engineering
Inha University, Incheon, Korea

Keywords: Large Eddy Simulation, Immersed Boundary Method, Buildings.

1 INTRODUCTION

Wind flow around buildings are of prime interest in wind engineering, and numerous studies in computations and experiments have been performed for flow around single building. Since Castro and Robins' experiments[1] of pressure on a cube, the investigation has deepening for the flow around a cube, and LES and DES have been tried in computational works.

However, only few of studies are found for the investigation of flow around a building complex, most of which have been done by experiments. This paper presents a large eddy simulation of turbulent flow around a building complex using the immersed boundary method (IBM). Due to high Re of the wind field, Lagrangian Dynamic Subgrid-scale model [2] is employed and wall model is applied on the surface instead of no-slip. The complicated geometry of the building complex does not pose any computational issues since IBM [3] allows for usage of Cartesian grids with forcing terms in the governing equations.

2 FORMULATIONS

2.1 Large Eddy Simulation

Based on the immersed boundary method and the mass forcing for continuity by Kim et al. [3], the governing equations for incompressible fluid flow using IBM and LES are as follows;

$$\frac{\partial \overline{u}_j}{\partial x_j} - q = 0 \quad \text{and} \quad \frac{\partial \overline{u}_i}{\partial t} + \frac{\partial \overline{u}_i \overline{u}_j}{\partial t} = -\frac{\partial p}{\partial x_i} - \frac{\partial \tau_{ij}}{\partial x_i} + \frac{1}{\text{Re}} \frac{\partial^2 \overline{u}_i}{\partial x_i \partial x_j} + f_i \qquad (1)$$

where x_i and $\overline{u}_i$ denote Cartesian coordinate and velocity component, respectively. p represents pressure, q and f_i denote mass source and momentum forcing, respectively. τ_{ij} is the Reynolds stress to be modeled. Re is Reynolds number based on reference length, h and inflow velocity U .

This study employs the Smagorinsky model based on the eddy viscosity.

$$\tau_{ij} = -2\nu_{SGS}\overline{S}_{ij} . \tag{2}$$

where ν_{SGS} is the eddy viscosity for LES which has the form of

$$\nu_{SGS} = \left(C_S\Delta\right)^2 \sqrt{2\overline{S}_{ij}\overline{S}_{ij}} \quad \text{and} \quad \overline{S}_{ij} = \frac{1}{2}\left(\frac{\partial \overline{u}_i}{\partial x_j} + \frac{\partial \overline{u}_j}{\partial x_i}\right) \tag{3}$$

Germano[4] suggested Dynamic Subgrid-scale model in which C_S is dynamically determined using averaging in the homogenous direction. However, the flow around a building complex does not present such a direction, which results in numerical instability in evaluating C_S. Instead, this study employs Lagrangian Dynamic Subgrid-scale model [2] which averages C_S in the paths of fluid particles. This results from minimizing the error of Germano's identity.

2.2   Immersed Boundary Method

The governing equations are discretized by a finite-volume method which has the second-order accuracy in space. The time integration is carried out by a fractional step method in which convection terms are integrated by a 3rd order Runge-Kutta method and diffusion terms are by Crank-Nicolson scheme.

$$\frac{\hat{u}_i^k - u_i^{k-1}}{\Delta t} = \left(\alpha_k + \beta_k\right)L\left(u_i^{k-1}\right) + \beta_k L\left(\hat{u}_i^k - u_i^{k-1}\right) - \gamma_k N\left(u_i^{k-1}\right) - \zeta_k N\left(u_i^{k-2}\right) - \left(\alpha_k + \beta_k\right)\frac{\partial p^{k-1}}{\partial x_i} + f_i \tag{4}$$

$$\frac{u_i^k - \hat{u}_i^k}{\Delta t} = -\frac{\partial \phi^k}{\partial x_i} \tag{5}$$

where L and N denote diffusion and convection operators, respectively, and the coefficients used in Eq. (4) are in Ref. [3].

In order to evaluate the momentum forcing f_i in Eq. (4), approximation of Eq. (1) is made by using a 3rd order Runge-Kutta method for convection term and a forward Euler method for diffusion terms;

$$\frac{U_i^k - u_i^{k-1}}{\Delta t} = \left(\alpha_k + \beta_k\right)L\left(u_i^{k-1}\right) - \gamma_k N\left(u_i^{k-1}\right) - \zeta_k N\left(u_i^{k-2}\right) - \left(\alpha_k + \beta_k\right)\frac{\partial p^{k-1}}{\partial x_i} + f_i \tag{5}$$

where U_i^k is the velocity inside the body, to be determined by the interpolation scheme described above. Rearranging Eq. (5) leads to the momentum forcing f_i which is used in Eq. (4).

3   COMPUTATIONAL RESULTS

3.1   Flow Around Wall-Mounted Single Cube

In order to verify the present method, turbulent flow around wall-mounted single cube is simulated. Inflow turbulence is imposed using random number to fit the longitudinal turbulence intensity given by a experiment. The number of grid is 192x128x96 and the results are compared with those of DES.

Fig (1) compares the present streamlines in vertical plane with those of DES [5], which are very similar to each other. Comparisons of the mean pressure in Fig (2) show that the present results are closer to the measurements.
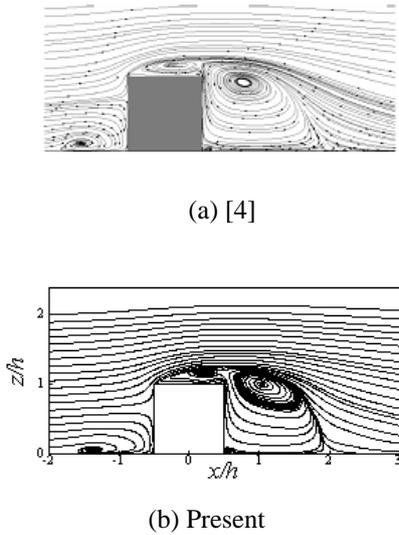

(a) [4]

(b) Present
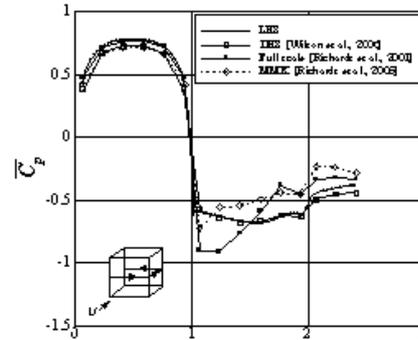
Fig. 1 Streamlines in the vertical center plane

Fig. 2 Pressure coefficients

3.2   Flow Around a Building Complex

The building complex modeled in this study was from Goettinger Strasse, Hanover, Germany [6] shown in Fig (3). The domain is 1320m, 1640m and 300m in x,y and z direction, respectively, along which 160x160x48 grid is used. Among 31 buildings, a building of 30m high is the tallest.
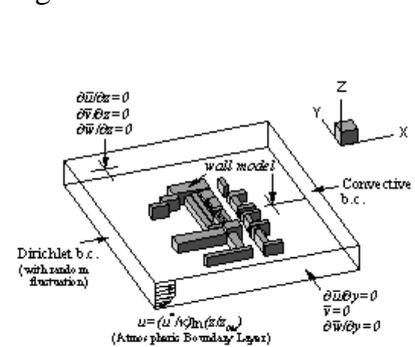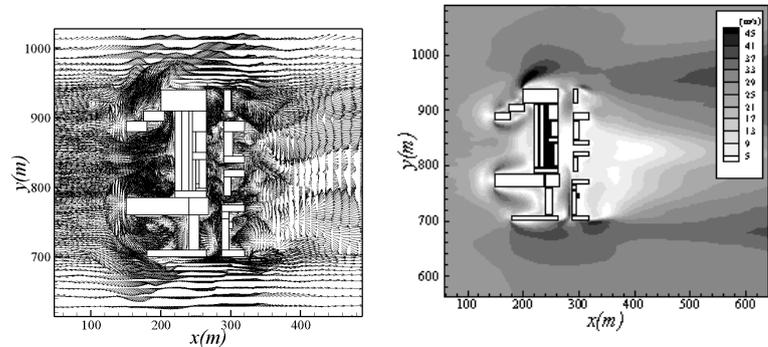

Fig. 1 Computational domain

Fig. 2 Instantaneous and Averaged Velocity Field

Fig (2) shows the instantaneous velocity field and the averaged magnitude of velocity at the height of pedestrian. The present computations well simulate the horseshoe vortex formed in front of the windward walls as well as the increase wind velocity through urban canyons. Fig (3) shows RMS of fluctuating pressure at the height of pedestrian which may be of importance to the safety of pedestrians as well as the generation of wind-borne debris. The results clearly demonstrates the possible hazard in some regions around the complex.

4   CONCLUSION

A practical method to simulate the wind flow of high Re around buildings has long been pursued in the wind engineering community. Hindered by computational capacity and/or

numerical difficulties, wind flow around a building complex has been studied mostly by experiments.

This study presents a computational method for wind flow around a building complex. LES is employed to simulate the wind flow of high Re with wall function. Conflicts between complicated geometry and grid are resolved by using IBM. Well-known problem of flow around wall-mounted single cube demonstrates the validity of the present method based on Lagrangian Dynamic Subgrid-scale model. A building complex of 31 buildings is modeled and wind flow is simulated. The present method shows that wind flow around the complex can be well predicted and it can be used on practical purposes.
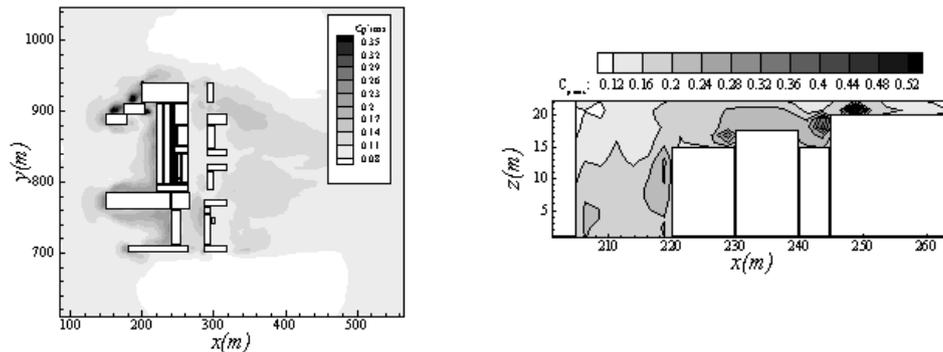

Fig. 1 RMS of fluctuating pressure on horizontal and vertical planes

ACKNOWLEDGEMENTS

REFERENCES

［1］ I. P. Casto, I. P and Robins A. G., The Flow Around a Surface-mounted Cube in Uniform and Turbulent Streams, *J. Fluid Mech.*, Vol. 79, part 2, 1977, pp. 307-335.

［2］ C. Meneveau, T.S. Lund and W.H. Cabot, A Lagrangian Dynamic Subgird-scale Model of Turbulence, *J. Fluid Mech.*, Vol. 319, 1996, pp. 353-385.

［3］ J. Kim, D. Kim and H. Choi, An Immersed-Boundary Finite-Volume Method for Simulations of flow in Complex Geometries, *J. Comp. Ph*ys., Vol. 171, 2001, pp. 132-150.

［4］ M. Germano, U. Piomelli, P. Moin and W.H. Cabot, A Dynamic Subgrid-Scale Eddy Viscosity Model, *Phys. Fluids*, Vol. 3, 1990, pp. 1760-1765.

［5］ R.P. Wilson, S.E. Haupt, L.J. Peltier and R.F. Kunz, Detached Eddy Simulation of Atmospheric Flow about a Surface Mounted Cube at High Reynolds Number, Proceedings of FEDSM2006, 2006, July 17-20, Miami, FL.

［6］ P. Louka, M. Ketzel, P. Sahm, E. Guilloteau, E., Moussiopoulos, N., Sini, J. -F., Mestayer, P. G. and Berkowicz, R., CFD Intercomparison Exercise within TRAPOS European Research Network", The 7th Conference on Environmental Science and Technology, Ermoupolis, Syros, Greece. 2001, pp. 267-275.