NUMERICAL SIMULATION OF TURBULENT FLOW AROUND A BUILDING COMPLEX

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

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

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

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

Abstract. *Strong wind flow around a building complex was numerically studied by LES. The original motivation of this work stemmed from the efforts to develop a risk assessment technique for windstorm hazards. Lagrangian-averaged scale-invariant dynamic subgrid-scale model was used for turbulence modeling, and a log-law-based wall model was employed on all the solid surfaces including the ground and the surface of buildings to replace the no-slip condition. The shape of buildings was implemented on the Cartesian grid system by an immersed boundary method. Key flow quantities for the risk assessment such as mean and RMS values of pressure on the surface of the selected buildings are presented. In addition, characteristics of the velocity field at some selected locations vital to safety of human beings are also reported.*

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;

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

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 based on algebraic identity between resolvable scale and subgrid scale, i.e.,

$$L_{ij} = \overline{\overline{u}_i \overline{u}_j} - \overline{\overline{u}_i}\,\overline{\overline{u}_j} \tag{4}$$

According to Lilly[5],

$$C_S^2 = \frac{L_{ij}M_{ij}}{M_{ij}M_{ij}} \tag{5}$$

$$M_{ij} = 2\Delta^2 \left(\overline{|\overline{S}|\overline{S}_{ij}} - 4\overline{|\overline{S}|}\,\overline{\overline{S}}_{ij}\right) \tag{6}$$

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.

The Lagrangian expression of the error and the accumulated error along the streamline are

$$e_{ij}(z,t') = L_{ij}(z,t') - C_S^2(x,t')M_{ij}(z,t') \tag{4}$$

$$E = \int_{-\infty}^{t} e_{ij}(z(t'),t')e_{ij}(z(t'),t')W(t-t')dt' \tag{5}$$

where $W(t-t')$ is a weight function and $z(t')$ is the previous position of fluid particle. Therefore C_S^2 with minimized E is

$$C_S^2 = \frac{\Re_{LM}}{\Re_{MM}} \tag{6}$$

where

$$\Re_{LM} = \int_{-\infty}^{t} L_{ij}M_{ij}(z(t'),t')W(t-t')dt'$$

$$\Re_{MM} = \int_{-\infty}^{t} M_{ij}M_{ij}(z(t'),t')W(t-t')dt' \tag{7}$$

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.

$$\frac{\hat{u}_i^k - u_i^{k-1}}{\Delta t} = (\alpha_k + \beta_k)L(u_i^{k-1}) + \beta_k L(\hat{u}_i^k - u_i^{k-1}) - \gamma_k N(u_i^{k-1}) - \zeta_k N(u_i^{k-2}) - (\alpha_k + \beta_k)\frac{\partial p^{k-1}}{\partial x_i} + f_i \qquad (8)$$

$$\frac{u_i^k - \hat{u}_i^k}{\Delta t} = -\frac{\partial \phi^k}{\partial x_i} \qquad (9)$$

where L and N denote diffusion and convection operators, respectively, and the coefficients used in Eq. (8) are in Ref. [3].

In order to evaluate the momentum forcing f_i in Eq. (8), 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;

$$\frac{U_i^k - u_i^{k-1}}{\Delta t} = (\alpha_k + \beta_k)L(u_i^{k-1}) - \gamma_k N(u_i^{k-1}) - \zeta_k N(u_i^{k-2}) - (\alpha_k + \beta_k)\frac{\partial p^{k-1}}{\partial x_i} + f_i \qquad (10)$$

where U_i^k is the velocity inside the body, to be determined by the interpolation scheme described above. Rearranging Eq. (10) leads to the momentum forcing f_i which is used in Eq. (8),

2.3   Wall Model

In LES for atmospheric boundary layer near wall boundaries at high Reynolds number, wall model in Eq. (11) can be utilized with the idea of average in the homogeneous direction.

$$\left\langle \tau_w^{\log} \right\rangle = -\left[\frac{\kappa}{\log(z / z_o)} \right]^2 \left\langle \overline{u}_1 \right\rangle^2 \qquad (11)$$

where κ is von Karman constant.

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 [6], which are very similar to each other. Horseshoe vortex forming on the windward surface is very similar to each other. Comparisons of the mean pressure in Fig (2) show that the present results are closer to the measurements.
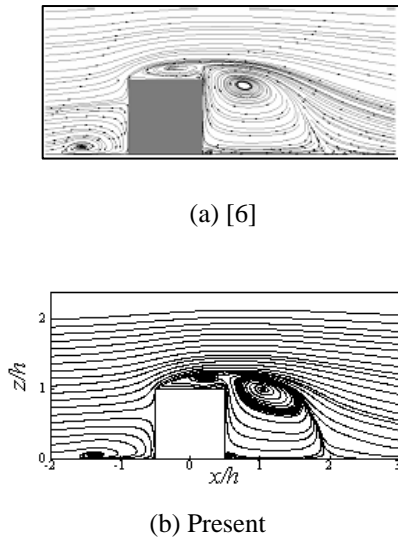
(a) [6]

(b) Present

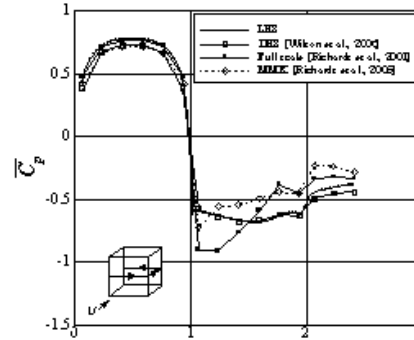
Figure. 1 Streamlines in the vertical center plane

Figure. 2 Pressure coefficients

3.2   Flow Around a Building Complex

The building complex modeled in this study was from Goettinger Strasse, Hanover, Germany [7] 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, No. 4 building of 30m high is the tallest and the geometry of all buildings is from the study of Louka et al [7]. Details of the buildings are listed in Table 1.
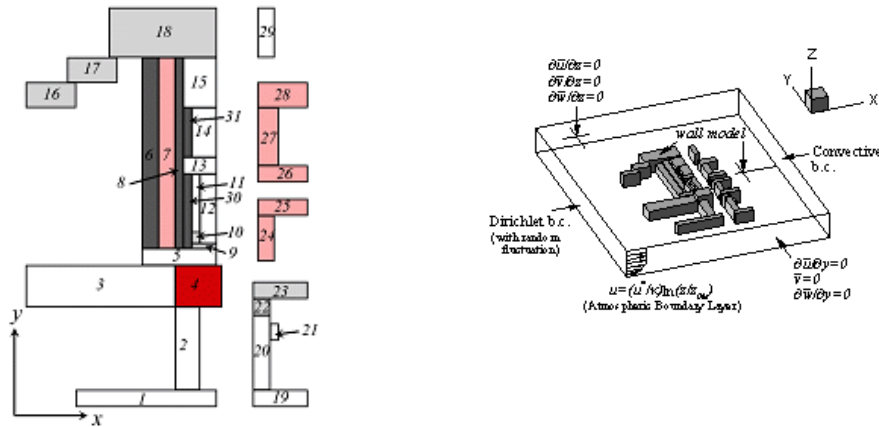

Figure. 3 Geometry of buildings and computational domain

As shown in Fig (3), Dirichlet and convective conditions are used on inlet and outlet, respectively, while slip condition is imposed on lateral and upper boundaries. On the inlet, atmospheric boundary layer profile is given as

$$u(z) = \frac{u^*}{\kappa} \ln(z / z_o) \qquad z < 100m$$
$$u(z) = 59.41 m / s \qquad z \geq 100m$$
(12)

4

where friction velocity $u^* = 3.1265$ and roughness length $z_o = 0.05$ are used. In addition, randomly generated turbulence is included on the inlet following Spalart [8] such that the turbulence intensity at the height of 30m matches 0.25 described in Ref. [7]. On the walls on buildings and ground, wall model is used with $z_o = 0.01$ as in Ref. [7].

Table 1. Details of buildings

No.	Length(x)	Width(y)	Height(z)	No.	Length(x)	Width(y)	Height(z)
1	180.0 ~ 265.0	700.0 ~ 710.0	20.0	17	175.0 ~ 205.0	895.0 ~ 910.0	23.0
2	240.0 ~ 255.0	710.0 ~ 760.0	20.0	18	200.0 ~ 265.0	910.0 ~ 940.0	23.0
3	150.0 ~ 240.0	760.0 ~ 785.0	20.0	19	287.0 ~ 320.0	700.0 ~ 710.0	20.0
4	240.0 ~ 268.0	760.0 ~ 785.0	30.0	20	287.0 ~ 297.5	710.0 ~ 755.0	20.0
5	220.0 ~ 265.0	785.0 ~ 795.0	20.0	21	297.5 ~ 298.0	740.0 ~ 750.0	20.0
6	220.0 ~ 230.0	795.0 ~ 910.0	15.0	22	287.0 ~ 297.5	755.0 ~ 765.0	23.0
7	230.0 ~ 240.0	795.0 ~ 910.0	17.5	23	287.0 ~ 320.0	765.0 ~ 775.0	23.0
8	240.0 ~ 245.0	795.0 ~ 910.0	15.0	24	290.0 ~ 300.0	787.5 ~ 815.0	17.5
9	250.0 ~ 265.0	795.0 ~ 798.3	20.0	25	290.0 ~ 320.0	815.0 ~ 825.0	17.5
10	250.0 ~ 255.0	798.3 ~ 805.0	23.0	26	290.0 ~ 320.0	835.0 ~ 845.0	17.5
11	250.0 ~ 255.0	805.0 ~ 840.0	20.0	27	290.0 ~ 302.5	845.0 ~ 880.0	17.5
12	255.0 ~ 265.0	798.3 ~ 840.0	20.0	28	290.0 ~ 320.0	880.0 ~ 895.0	17.5
13	245.0 ~ 265.0	840.0 ~ 850.0	20.0	29	290.0 ~ 300.0	910.0 ~ 940.0	20.0
14	250.0 ~ 265.0	850.0 ~ 880.0	20.0	30	245.0 ~ 250.0	795.0 ~ 840.0	15.0
15	245.0 ~ 265.0	880.0 ~ 910.0	20.0	31	245.0 ~ 250.0	850.0 ~ 880.0	15.0
16	150.0 ~ 180.0	880.0 ~ 895.0	23.0				

Fig (4) compares the present results with Louka et al [6] for the averaged 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 while Louka et al [6] did not.
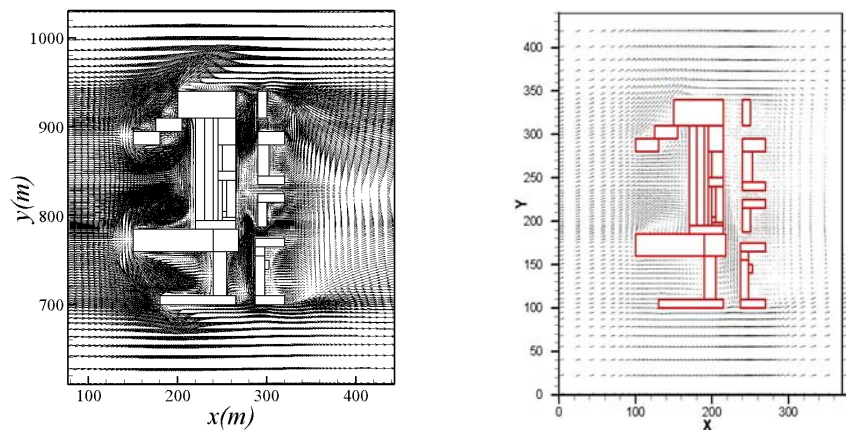

Figure 4. Comparison of averaged velocity field at pedestrian level (Present on left, Louka et al [7] on right)

Fig (5) shows the magnitude of horizontal velocity field at the height of pedestrian which may be of importance to the safety of pedestrians as well as the generation of wind-borne debris. It shows that the speeds are twice the inlet velocity in some region and very high along the aisle in front of No. 20 building. The vertical velocity in Fig (5) shows that strong upwind is formed around No. 16, 17 and 18 buildings resulting in strong vortex.
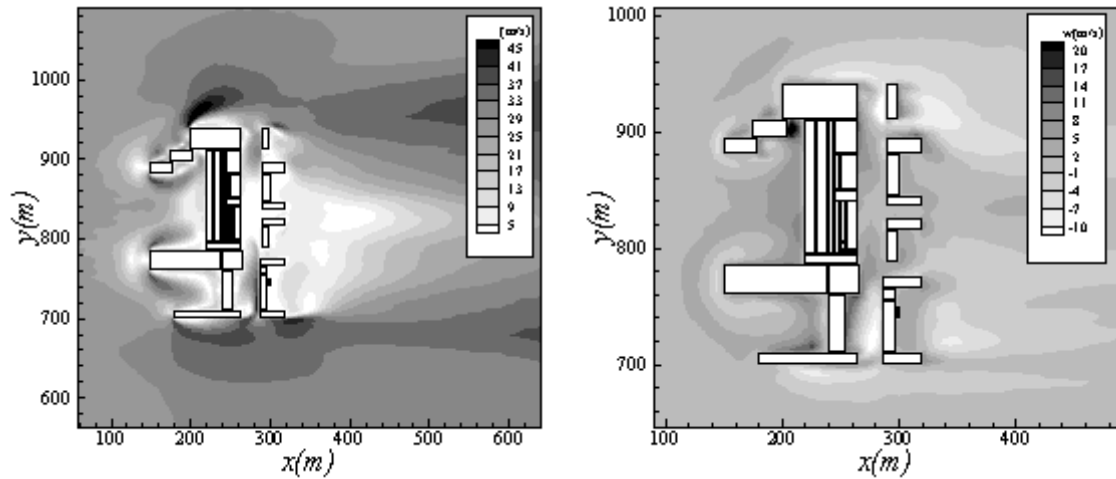
Figure 5. Averaged magnitude of horizontal speed (left) and vertical velocity (right) at z=1.5m

Fig (6) displays RMS of pressure distribution at z=1.5m, which shows the strong vortex around No. 16, 17 buildings results in large RMS of the fluctuating pressure. The right figure in Fig (6) shows RMS on the vertical plane around the region.
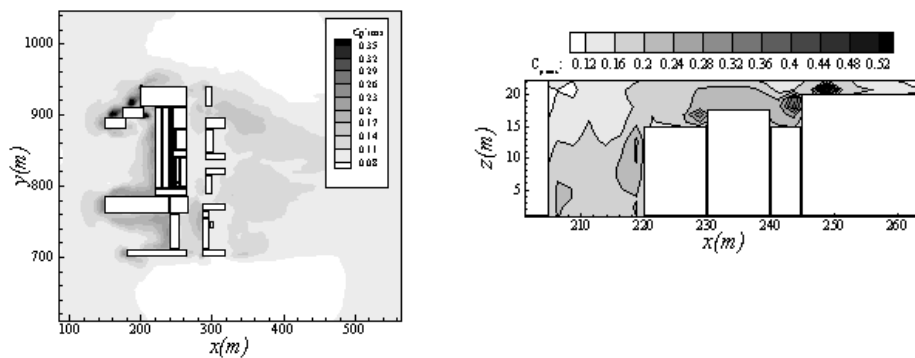
Figure 6. RMS of fluctuating pressure on horizontal and vertical planes

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.

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］ a Lilly, D. K., "A proposed modification of the Germano subgrid-scale closure method", Phys. Fluids, Vol. 4, 1992, pp. 633-635.

［6］ 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.

［7］ 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.

［8］ Spalart, P. R, "Direct Simulation of a Turbulent Boundary Layer up to Re=1410", J. Fluid Mech., Vol. 187, 1988, pp. 61-98.