Large-eddy simulation of flow around obstacle arrays using drag force method of gas-solid two-phase flow

Jianying Jiao a, Ryuichiro Yoshie b

aTokyo Polytechnic University, Kanagawa, Japan  
bTokyo Polytechnic University, Kanagawa, Japan

ABSTRACT: The traditional method (body-fitted method) of simulating flows around obstacle arrays is usually used to simulate urban environments. However, with increasing numbers of simulated buildings, large amounts of time and labor are required to generate body-fitted grids. In order to solve this problem, the drag force method using gas-solid two-phase flow was applied to simulate an urban environment. At first, it was important to assess the precision of the drag force method compared with that of the traditional method. Thus, numerical tests were carried out on a high-rise building and building arrays. The results obtained from the traditional method and the drag force method were in good agreement with experimental data. The grids for the drag force method didn’t need to be body-fitted, so it required much less time to generate them.

KEYWORDS: Large eddy simulation; subgrid-scale model; drag force.

1 INTRODUCTION

With the increase of urban populations, urban environments have become an important research subject. Increasing numbers of researchers are using CFD to study urban environments. Because of the increase of the computational domain of urban environments and simulated buildings, huge number of grids is needed and a great deal of time and labor are required to generate body-fitted grids. In order to solve this problem, it is not enough to simply improve computer performance. A better method for simulating an urban environment needs to be developed.

In the early development stage, restrictions of computer performance enabled flow fields around only simple geometry to be simulated using the traditional method. For example, the flow field around a square cylinder (or other simple geometry building) could be simulated using large eddy simulation. Kelkar and Patankar (1992) 1 predicted vortex shedding behind a square cylinder. Mukhopadhyay et al. (1992) 2 researched wakes behind a square cylinder in a channel. Rodi et al. (1997) 3 simulated the flow field around a square cylinder and a wall-mounted cube flow in a channel. Bouris and Bergeles (1998) 4 predicted vortex shedding from a square cylinder in 2D.

With development of computers, flow fields around building arrays could be simulated by the traditional method. Hanna et al. (2002) 5 used the Smagorinsky model to simulate the flow field around simple obstacle arrays. Tseng (2006) 6 predicted urban dispersion using large eddy simulation. Xie and Castro (2009) 7 applied LES to simulate flow and dispersion in urban streets. Gousseau et al. (2011) 8 adopted RANS and LES to simulate the wind field and pollution dispersion around a group of buildings in a downtown area.

With the increasing numbers of simulated buildings, a great deal of time is required to generate body-fitted grids. To overcome this problem, a drag force method of gas-solid two-phase flow is used to simulate urban environments. This method is introduced in Section 2. Section 3 compares numerical results obtained by this method for wind fields around a high-rise building and building arrays with those obtained by the traditional method and experiment data. Conclusions are given in Section 4.
2 DRAG FORCE METHOD

In our drag force method, the buildings are firstly discretized by a first set of grids, as shown in Fig. 1(a). This first set of grids is a body-fitted one, in which each grid in the building is assumed to be filled with a sphere, as shown in Fig. 1(b). The volume of the sphere is determined such that the air void fraction rate \((= (V_r-V_s)/V_r)\) becomes very small (for example 0.001), where \(V_r\) = volume of rectangular cell and \(V_s\) = volume of sphere.

Secondly, a second set of grids is generated, as shown in Fig. 1(c), in which black and red lines show the second and first set of grids, respectively. The second set of grids is not a body-fitted one and is used for flow calculations. Cells that contain solid parts of a building in the second set of grids are expressed by solid void fractions as:

\[
\varepsilon_p = \sum_{i=1}^{N} \frac{V_i}{(\Delta x \Delta y \Delta z)}
\]

(1)

where \(\Delta x\), \(\Delta y\), \(\Delta z\) are grid lengths of the second set of grids that contain solid parts of a building, \(N\) is the number of spheres in the second set of grids and \(V_i\) is the volume of the \(k^{th}\) sphere in the second set of grids.

The air void fraction is expressed as:

\[
\varepsilon_{\alpha} = 1 - \varepsilon_p
\]

(2)

The drag force coefficient determined by Ergun/Wen and Yu (1952; 1966)\(^{9,10}\) was used, and is expressed as:

\[
\beta = \frac{3}{4} \left(1 - \varepsilon_p\right) \varepsilon_{\alpha} \rho_s \left| u_s - u_p \right| + \frac{150 (1 - \varepsilon_p)^2 \mu_s}{\varepsilon_p \rho_s \Delta V} + \frac{175 (1 - \varepsilon_p) \rho_s \left| u_s - u_p \right|}{\varepsilon_p \Delta V}, \varepsilon_{\alpha} > 0.8; 150 \left(1 - \varepsilon_p\right) \mu_s \rho_s \left| u_s - u_p \right|, \varepsilon_{\alpha} \leq 0.8
\]

(3)

\[
C_{co} = \begin{cases} 
24(1 + 0.15 \text{Re}^{0.5}) / \text{Re}, & \text{Re} \leq 1000 \\
0.43, & \text{Re} > 1000
\end{cases}
\]

(4)

where \(u_s\), \(u_p\), \(\rho_s\) and \(\mu_s\) are gas velocity, solid velocity, gas density and gas viscosity coefficient, respectively. \(u_p\) is zero in this method.

The filter (LES) momentum equation is expressed as:

\[
\nabla \cdot \left[ (\varepsilon_{\alpha} u) \mu_s \right] = 0
\]

\[
\varepsilon_{\alpha} \frac{\partial u_s}{\partial t} + u_s \cdot \nabla u_s = -\frac{\varepsilon_{\alpha} \nabla p}{\rho_s} + \varepsilon_{\alpha} \nabla \cdot \left( \nabla u_s + \nabla u_s^T \right) - \frac{\beta u_s}{\rho_s}
\]

(5)
Even if the cells are totally inside the building (even the cells are totally solid), they are treated as fluid cells with very small \( e_g \) (for example, \( e_g = 0.001 \)). Thus, it becomes possible to universally apply equations (3) for every cell regardless of its phase (fluid or solid).

With regard to the numerical tests in this paper, the new subgrid-scale model (Gu and Jiao 2011)\textsuperscript{11} was used.

3 NUMERICAL TEST

3.1 HIGH-RISE BUILDING

Fig. 2 shows a high-rise building with the same geometry as that of the wind-tunnel experiment of Meng and Hibi (1998)\textsuperscript{12}. The Reynolds number based on \( H \) (building height) and \( U_H \) (inflow velocity at \( z = H \)) was 24,000. \( H, L \) and \( W \) were 0.16m, 0.08m and 0.08m. The flow field around the high-rise building was simulated by using the traditional method (body-fitted method) and the drag force method. The results by the two methods were compared with experiment data (Meng and Hibi 1998).

The grids and boundary conditions of the numerical test were the same as those of Tominaga et al.\textsuperscript{13} (2008). The non-uniform grid and the inflow condition of Kataoka (2002, 2008)\textsuperscript{14,15} were used. The grid numbers using the two methods were almost the same: 60×45×39. The velocity components at the inflow boundary (Kataoka 2008) are given by:

\[
\begin{align*}
    u_{\text{inlet}}(y, z, t) &= <u>_{\text{inlet}}(y) + \phi(\theta) \times [u(y, z, t) - <u>(y, z)]_{\text{recy}} \\
    v_{\text{inlet}} &= \phi(\theta) \times [v(y, z, t) - <v>(y, z)]_{\text{recy}} \\
    w_{\text{inlet}} &= \phi(\theta) \times [w(y, z, t) - <w>(y, z)]_{\text{recy}}
\end{align*}
\]

where subscripts denote the values at the inlet and the downstream (recy) planes and \(<->\) is a time-averaged value. \( \phi(\theta) \) is the damping function (Kataoka 2008).

Air void fraction in the high-rise building was set to 0.05 in the drag force method.

![Fig. 2 Geometry of high-rise building](image)

Figs. 3 and 4 show the mean velocity profiles and the r.m.s. value profiles at the \( y/b = 0 \) plane. There were nine profiles: at \( x/b = -0.75, x/b = -0.5, x/b = -0.25, x/b = 0, x/b = 0.5, x/b = 0.75, x/b = 1.25, x/b = 2 \) and \( x/b = 3.25 \). The mean velocities by the drag force method were a little closer to the experiment data than those by the traditional method, as shown in Fig. 3. The r.m.s. values by the traditional method and the drag force method were in agreement with the experiment data, as shown in Fig. 4.
**Fig. 3.** Mean velocity profiles

(a) Mean velocity profiles in streamwise direction \( U (y/b = 0) \)

(b) Mean velocity profiles in normal direction \( W (y/b = 0) \)

**Fig. 4.** r.m.s. value profiles

(a) r.m.s. value profiles in streamwise direction \( u_{rms} (y/b = 0) \)

(b) r.m.s. value profiles in spanwise direction \( v_{rms} (y/b = 0) \)

(c) r.m.s. value profiles in normal direction \( w_{rms} (y/b = 0) \)

**Figs. 5 and 6.** Mean velocity profiles at \( z/b = 0.125 \) and \( 1.25 \) plane, respectively. The numerical results in Figs. 5 and 6 are in agreement with the experimental data. The results by the drag force method are closer to experimental data than those by the traditional method.

**Fig. 5.** Mean velocity profiles

(a) Mean velocity profiles in streamwise direction \( U (z/b = 0.125) \)

(b) Mean velocity profiles in spanwise direction \( V (z/b = 0.125) \)
The major purpose for the numerical simulation of the high-rise building was to validate the precision of the drag force method. It was found that the results by the drag force method were closer to experimental data than those by the traditional method. The numerical results verify that the drag force method can be used to simulate flow field around a high-rise building.

3.2 BUILDING ARRAYS

The flow field around the building arrays was simulated using the traditional method and the drag force method. The geometries of the building arrays, as shown in Fig. 6, were the same as those in the wind-tunnel experiment of Davidson et al. (1996)\(^\text{16}\). The height \(H\), length \(B\) and width \(W\) were 0.12m. The grid number and boundary conditions were the same those of Gu and Jiao (2011)\(^\text{17}\). The non-uniform grids in the computational domain were used for the simulations of the turbulent flows around staggered and aligned obstacle arrays. The grid numbers of the traditional method and the drag force method were almost the same. The staggered obstacle array had about 816,000 grids and the aligned obstacle array had about 672,000 grids. The velocity formulation at the inflow boundary of Kataoka (2002, 2008) (equation (6)) was used. In the drag force method, the air void fraction in the obstacles was set to 0.05.

Figs. 7 and 8 show the numerical streamwise mean velocity profiles of the staggered obstacle array and the aligned obstacle array, which were compared with experimental data. It was found that the results of the traditional method and the drag force method were in agreement with experimental data.
The numerical simulation results by the drag force method and the traditional method of the staggered obstacle array and the aligned obstacle array were in agreement with experimental data. Thus, the drag force method could be applied to simulate an urban environment. The drag force method doesn’t need body-fitted grids and can thus reduce the time required to generate grids.

4 CONCLUSION

With the increasing numbers of simulated buildings required to research urban environments, preprocessing for grids takes a great deal of time and labor, especially when generating hexahedron body-fitted grids. To simplify the generation of grids, a drag force method of gas-solid two-phase flow was proposed. Numerical tests have shown that the results by the traditional method and the drag force method were in agreement with experimental data. The advantages of the drag force method are that it doesn’t need body-fitted grids and thus greatly simplifies grid generation, and that it also simplifies simulation of obstacle arrays.

5 ACKNOWLEDGEMENTS

This study was funded by the Ministry of Education, Culture, Sports, Science and Technology, Japan, through the Global Center of Excellence Program, 2008-2012, which is gratefully acknowledged.
6 REFERENCES

5. S. R. Hanna et al, Comparisons of model simulations with observations of mean flow and turbulence within simple obstacle arrays, Atmos Environ 36 (2002) 5067–5079