LARGE EDDY SIMULATION ON BUILDING AERODYNAMICS

Tetsuro Tamura

Tokyo Institute of Technology, Yokohama, Japan
tamura@depe.titech.ac.jp

ABSTRACT:
This is a review paper addressing the Large Eddy Simulation (LES) on estimation of wind effects on buildings and structures. First, applicability and effectiveness of the sub-grid scale models which have been proposed recently, to turbulent wind accompanied with a massive flow separation in a building wake are summarized by surveying the previous research results. Also, the external forcing technique based on the immersed boundary concept to represent various shapes of buildings and structure as boundary conditions is explained in order to simply realize LES of the turbulent flows around a very complicated configuration. Turbulent flows and pressure distributions around a bluff cylinder with a simple sectional shape are given as fundamental numerical examples in the present study. Also, with regard to a tall building with actual shape existing among densely arrayed other tall buildings in a city, the aerodynamic characteristics such as a surrounding wind, wind pressures and wind forces are examined by LES as an application in structural wind engineering, and furthermore their predictive accuracy has been discussed in comparison with the field measurement data as well as wind tunnel experimental data.

KEYWORDS: STRONG WIND, LES, WIND PRESSURE, WIND FORCE, AERODYNAMICS

Introduction
Due to recent distinctive development of various techniques for computational wind engineering which has been thus far mainly used in view of the researches for understanding some kind of physical mechanism, we are now approaching the stage to employ these techniques for solving the unresolved practical problems in the designing process of wind engineering. Especially, in order to evaluate the wind resistant performance of buildings and structures, the unsteady time-dependent analysis such as the large eddy simulation (LES) is indispensable and its potential for predictive accuracy in structural design under the strong wind should be clarified. This study addresses the LES technique on estimation of wind effects on buildings and structures. Applicability and effectiveness of the sub-grid scale models which have been proposed recently, to turbulent wind accompanied with a massive flow separation in the building wake are discussed by surveying the previous research results. Also, the external forcing technique based on the immersed boundary concept to represent various shapes of buildings and structure as boundary conditions in the numerical analysis of wind-related problems is explained in order to realize simply LES of the turbulent flows around a very complicated configuration. Turbulent flows around a two-dimensional circular and rectangular cylinder in uniform flow very at high Reynolds numbers, or around a three-dimensional square cylinder in boundary layer are given as fundamental numerical examples...
in the present study. Physical mechanism of pressure fluctuations including the peak values on the cylinder-type structure is examined by analyzing the computed data previously obtained by LES. Also, with regard to the flow fields in the urban canopy formed by densely-arrayed buildings with actual sectional shapes and various heights, the predictive accuracy of LES for such a complicated configuration has been investigated in comparison with wind tunnel experimental data. Wind velocities, wind pressures and wind forces on a tall building in actual urban area have been provided as a numerical application of the LES coupled with the meteorological model and are investigated for classifying the resulting aerodynamic characteristics subject to various conditions, such as the building configuration, its surrounding objects or the influential meteorological events etc. Generally the computed data have been validated by comparing with the full-scale measurement data as well as the wind tunnel experimental data. Especially in this paper, the effects of the circumference environment consisting of other tall buildings and the location and the direction of adjacent streets on aerodynamic behavior of the specified tall building under strong wind are elucidated in detail.

**Current state of LES researches**

**Sophisticated SGS models for LES**

Here we explain new type of sub-grid scale (SGS) models for LES. The most conventional SGS model in the past is the Smagorinsky model which has been deduced by the eddy viscosity concept. In this model the governing equations consist of the filtered forms of the continuity equation and the Navier-Stokes equation as follows:

\[
\begin{align*}
\frac{\partial \tilde{u}_i}{\partial t} + u_j \frac{\partial \tilde{u}_i}{\partial x_j} &= -\frac{1}{\rho} \frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j} \left( -\tau_{ij} + 2\nu S_{ij} \right) \\
\tau_{ij} &= u_i u_j - u_j u_i, \quad S_{ij} = \frac{1}{2} \left( \frac{\partial \tilde{u}_i}{\partial x_j} + \frac{\partial \tilde{u}_j}{\partial x_i} \right) \\
\tau_{ij} - \frac{1}{3} \delta_{ij} \tau_{kk} &= -2\nu \nu_e S_{ij} \\
\nu_e &= (C_s \Delta)^2 \left( 2S_{ij} S_{ij} \right)^{1/2}
\end{align*}
\]

where \( i=1,2,3 \) correspond to the directions \( x, y \) and \( z \), respectively. The summation rule is assumed for repeated indices. \( u_i, p, t, \tau_{ij} \) and \( \nu \) denote velocity, pressure, time, the SGS Reynolds stress and molecular viscosity coefficient, respectively. The Smagorinsky model is subject to the eddy viscosity concept and the SGS Reynolds stress is presented using the strain rate tensor \( S_{ij} \) and the eddy viscosity coefficient \( \nu_e \). \( Cs \) of the eddy viscosity is given by the constant value in the standard model but determined instantaneously based by the Geomano identity in the dynamic model (Geomano et al., 1991). For the effective performance of this model, the numerical error analysis has recommended the spatial derivatives to be approximated by more than the fourth-order central difference.

The Smagorinsky model requires the assumption of the equilibrium mainly for two quantities of the production and the dissipation of turbulence energy, so good and reasonable results have been obtained for the boundary-layer type of turbulent flows by LES. However the separated flows often encounter in wind engineering cannot be simulated with good
accuracy by the standard type of Smagorinsky model where the value of $C_s$ is constant for the flow field. Especially for the complicated and very unsteady flows around a bluff body, the massive separation is accompanied and the universality of this constant for various types of flow structures has not been ensured.

The advent of the Geomano identity has made it possible to estimate this constant value using the filtering operation to the turbulent flow field. This is the dynamic Smagorinsky model where the dynamic procedure is applied for the Smagorinsky model and the constant value obtained is changed spatially and temporally, so determined as a relatively stable value by the averaging technique based on the homogeneity of turbulent flow field. Afterwards, almost turbulent flows appearing in wind engineering can be simulated with sufficient accuracy using the dynamic Smagorinsky model. As a demerit of this model, the numerical stability cannot be obtained easily because the constant is not realized to be always stable sufficiently and sometimes becomes a negative value. So, actual computation is not necessarily feasible and generally difficult. Also, the dynamic Smagorinsky model is still a kind of the eddy viscosity model which cannot generate a good result for the impinging flows and the secondary flows with swirling. The mixed dynamic model (Zang et al., 1993) with the scale similarity concept was proposed and has solved the above limitations.

Recently a new type of SGS model has been developed based the structure function (Kobayashi et al., 2008). This model is an eddy viscosity model having a functional constant value, and does not require the dynamic procedure and the damping function near the wall. This model has realized easy and user-friendly numerical procedure. The mixed-time-scale SGS model is also using fixed model parameters (Inagaki et al., 2005). Besides, there is a SGS model where the model constant is dynamically determined as one value for the whole computational region (You and Moin, 2007).

**Immersed boundary method**

The shape of a building is a significant factor that affects the behavior of the shear layer separating from the building and the vortex formation in the wake. These flow structures relate strongly to the wind loads on buildings. With regard to the actual circumstance in the urban area, many buildings are built very-complicatedly. So it is required to reproduce the shape and the surface roughness of buildings appropriately enough to simulate the flow even under insufficient grid resolution. To solve this problem, the Immersed Boundary Method (IBM) is expected, which uses boundary body forces that impose the boundary conditions on a given surface, not coinciding with the grid lines.

For the IBM, a body inside flow fields is represented as body forces in the Navier-Stokes equations. The IBM has been widely developed since the early work by Peskin (1972). Mohd-Yusof (1996) provided the discrete-time body force in a spectral method, which does not require smaller computational time step compared to previous methods. After that, Fadlun et al. (2000) has extended this approach to a finite difference method on a Cartesian staggered grid and verified the accuracy for various types of complicated flows.

The discrete-time body force by Fadlun et al. (2000) can satisfy the no-slip condition on the immersed boundary as follows. The time-discretized version of the Navier-Stokes equations can be given by

$$\frac{\mathbf{u}_i^{n+1} - \mathbf{u}_i^n}{\Delta t} = \text{RHS}^{n+1/2} + f_i^{n+1/2}$$

$$\text{RHS}^{n+1/2} = -\frac{1}{\rho} \frac{\partial}{\partial x_j} (\mathbf{u}_i \cdot \mathbf{u}_j) v - \frac{\partial p^n}{\partial x_j} + \nu \frac{\partial^2 u_i^n}{\partial x_j \partial x_j}$$
where $\Delta t$ is the computational time step and the superscript denotes the time step number. To impose $\vec{u}_i^{n+1} = \vec{V}_b$ on an immersed boundary, the body force $f_i^{n+1/2}$ must be written as

$$f_i^{n+1/2} = -\text{RHS}^{n+1/2} + \frac{V_k - \bar{u}_i^n}{\Delta t}$$

where $f_i^{n+1/2}$ can be obtained according to the location of grid point. For the grid points inside the body the value of $\vec{V}_b$ in the right hand side is also given to be zero. For the firstly adjacent grid point external to the body, $\vec{V}_b$ is obtained by linear interpolation between the velocity at the body surface and the velocity at the second grid point outside of the body. For the other grid points, $f_i^{n+1/2}$ is given to be zero.

Tamura and Kono et al. (2003) has applied this IBM technique to turbulent flows among multiple tall buildings. Figure 1 depicts the instantaneous vorticity contours around three actual tall buildings. We can certainly recognize the vortex structures which are much deformed in the complicated arrangement of these three tall buildings. It can be presumed that the wind flow, is coming between two upstream tall buildings, increases its velocity and impinges on the frontal surface of a downstream tall building. Also, strong horse-shoe type vortex can be seen at the bottom of the tall building. According to details of flow patterns, the separated shear layer from the upper corner of building (A) comes along the side of the building (A), also being affected by the wall of building (B). The wake flow of building (A) is trapped in the frontal region of building (C). As a result, the pressure distribution on the frontal surface of building (C) is largely deformed, as seen in Figure 2. We can also recognize an interaction of shear layers separated from other corners of different buildings. As a result of vortices weaken and moved away from the wall, the pressure distribution on the surface (IV)-(I) of building (C) indicates the slight recovery. A strong horse-shoe type vortex at the bottom of the building (C) has a much influence on surrounding wind environment. Wind load is also influenced by the adjacent buildings and the frontal pressure in lower region decreases compared to the case of a single building.

**Generation of inflow turbulence**

In many cases of LES for wind engineering, generation of inflow turbulence, which provides time sequential velocity data with physically-corrected spatial structures, is a very important issue, because wind is essentially a turbulent and unsteady flow at very high Reynolds numbers. Thus, the oncoming flow for the computational domain should have these inherent characteristics of natural wind. One method for obtaining this type of flow is to numerically simulate turbulent flows in the auxiliary computational domain (often called a driver region where a fluid flow is driven). Lund et al. (1998) proposed the technique of inflow generation, where various development ratios of boundary-layer parameters such as depth and friction velocity are estimated using the computed result just obtained sequentially. The velocity profile at the inflow boundary is rescaled based on the law of the wall or defect law assuming the spatial development of a boundary layer on a smooth surface. Nozawa and Tamura (2002) extended Lund’s method to a rough-wall boundary layer flow where the length scale of rough wall is estimated based on roughness density and the experimental formula for rough-wall friction is used for predicting the development of a boundary layer over a rough surface.

Another method is to generate inflow turbulence statistically. This method is mainly employed for homogeneous turbulence as a two-dimensional problem, such as flow around a cylinder-type structure. However, statistically generated inflow turbulence might not satisfy the governing equations for incompressible flows, thus requiring velocity correction. This usually requires a very large amount of storage for time-sequential data.
Fundamental issues on flows around a two-dimensional bluff cylinder

**LES of flows around a circular cylinder using the overset grid system**

In order to carry out LES of turbulent flows around various buildings with complex geometry, it is generally effective to introduce an overset method whose grid system is composed of more than two component grids. Then, a kind of interpolation of physical quantity is required for connecting component grids in fluid flows. But the interpolation interferes with the conservation of numerical schemes, which is required for simulating fluctuations in turbulent flow.

Itoh and Tamura (2006) have provided the overset grid method for LES of the flows around a circular cylinder. For the high Reynolds number flows, the artificial numerical viscosity has been still utilized for LES of turbulent flows around a bluff body. Taking into consideration this situation, they have evaluated effects of the numerical viscosity on the flow structures and the aerodynamic characteristics for a circular cylinder. Also, for the computed results obtained by using the overset grid method, some wiggles often appear in the pressure field at overset region due to the error in the interpolation process. They have proposed a new method to remove this error in the overset grid system.

For numerical scheme of the convection term, the higher order interpolation method in skew-symmetric form is used as follows:
\[
\frac{1}{2} \left( JU \frac{\partial f}{\partial \xi} + \frac{\partial JU f}{\partial \xi} \right) = \frac{1}{2} \left[ JU \left( \frac{\partial f}{\partial \xi} + \frac{\partial JU f}{\partial \xi} \right) \right] + \alpha JU \left( f_{i-2} - 4f_{i-1} + 6f_i - 4f_{i+1} + f_{i+2} \right) \frac{12\Delta \xi}{f_{i-2} - 27f_{i-3/2} + 27f_{i+1/2} - f_{i+3/2}} \right).
\]

where, in order to avoid numerical instability of a convection term, the 4th-order numerical dissipation has been added with \( \alpha = 0.5 \), which is a half of value used in the UTOPIA scheme.

Figure 3 illustrates the overset grid system for a circular cylinder. For the whole computational domain, non-uniform Cartesian grid is used, while the O-type of body-fitted grid is surrounded for the region attached to a circular cylinder. According to vortical structures around a circular cylinder represented by the instantaneous vorticity contours in Figure 4, deformation in the spanwise direction of a separated shear layer is smaller and fine structures uniformly spread in the case of the central difference scheme. The case of upwind scheme shows longitudinal vortices wrapped around the Karman type of vortices. Figure 5 shows the effect of the numerical dissipation on the mean pressures on a circular cylinder. The central difference case shows good agreement with the experimental and DNS (direct numerical simulation) data. The upwind case shows smaller values of mean pressure at back of a circular cylinder. This is because the returning flow to the back becomes weak.

In the process of solving the pressure Poisson equation, the numerical error induced by the interpolation procedures of overset grid method, causes the discrepancy of continuity equation, which directly influences the accuracy of pressure. Itoh and Tamura (2006, 2007) proposed the correction method of mass conservation procedures where the numerical discrepancy of a continuity equation is resolved introducing a source term of pressure correction for the Poisson equation of pressure. This new overset grid method is expected to improve the velocity field as a result of avoiding the inconsistency of the continuity equation along the boundary of the overlapping grid.

Figure 6 shows the effectiveness of a new solver for pressures on a circular cylinder. For the mean pressure coefficients, the new and the original method show almost same results. But for the fluctuating pressure coefficients, the results obtained by the new method shows larger values compared to the results by the original method and are consistent with the experimental data. Surely the new pressure solver of the overset method has worked well for accurate numerical results of aerodynamic characteristics.

**Applicability of LES to the drag crisis of a circular cylinder**

Ono and Tamura (2008) carried out LES of flows around a circular cylinder in the super critical Reynolds number region and discussed the applicability of the LES method to the separated shear layer accompanied with turbulence transition sensitively occurring at very high Reynolds number. There have been many researches on the applicability of various numerical models to this situation. However, almost all previous computations have shown an ambiguous decrease in the drag coefficient and an insufficient increase in vortex shedding frequency in comparison with the experiment. Furthermore, the details of the complicated flow structure in the critical region have never been captured by numerical approach. In this research, a sophisticated LES has been systematically performed under various numerical conditions. The flow around a circular cylinder at very high Reynolds number is investigated using the LES method, focusing on the complicated flow structure near flow separation point.

The over-set grid method is employed to distribute the grid points over the whole domain around a circular cylinder with a balanced size. Figure 7 shows the overset grid system near the cylinder. It is composed of three kinds of meshes. The Reynolds number is
equal to 600,000. The dynamic mixed model is used to study the effect of the SGS model.

With regard to the effects of the numerical dissipation, it is shown that a large numerical dissipation ($\alpha=0.2$) produce the results with higher values of time-averaged drag coefficient than the other cases. When using $\alpha$ smaller than 0.05, the computations became unstable and blown up. In the numerical conditions used, the calculations using finer grid resolution in the span-wise direction succeed in simulating a rapid tendency of an increase in the Strouhal number and of a decrease in the time-averaged drag coefficient, observed in the previous experiments.

Figure 8 depicts the contours of stream-wise velocities around a circular cylinder. The very narrow near wake, which results in sufficient values for a decrease in $C_D$ and an increase in $St$, is recognized. The computed velocity vectors near the separation point for a circular cylinder are also shown in Figure 9. It can be seen that the flow separates from the cylinder surface and then reattaches. These flow characteristics cause the separation bubble to be instantaneously formed. Looking at the details of the separation bubble, a secondary vortex that has positive velocities is recognized. It should be noted that this secondary vortex causes a sensitive property such as the skin friction to slightly increase after the flow separation.

**LES of flows around a rectangular cylinder**

Itoh and Tamura (2007) carried out the large eddy simulation of turbulent flow around a rectangular cylinder with side ratios of 1.0, 2.0, 2.67, and 3.0 at $Re = 22000$, using an overset grid system. Based on the new pressure solver, the aerodynamic forces of rectangular cylinders can be numerically predicted with sufficient accuracy. Details of pressure distributions along the side surface of the cylinder are discussed as well. The applicability of LES for the aerodynamic problems of a rectangular cylinder with various side ratios has been elucidated.

In this study, the turbulent flows around a rectangular cylinder with various side ratios in uniform flow have been discussed. The computational domain consists of both the Cartesian grid system and the O-type boundary-fitted grid system in Figure 10. The former covers an entire computational domain that makes it possible to ignore blockage and wall effects. The latter is placed only around a rectangular cylinder that can reproduce an earlier boundary layer near the stagnation point of a front surface, as well as the separated shear layers from the leading edge of the cylinder. Definitions for a rectangular cylinder model are given Figure 11.

The statistics of aerodynamic characteristics have been compared with the previous experimental results in Figure 12. The Strouhal numbers obtained by time variation in lift and the time-averaged drag are in reasonably good agreement with the experimental results. There are two Strouhal numbers when $D/B = 2.67$, but there is only one Strouhal number when $D/B = 3.0$. Small differences can be seen in the standard deviation of drag and lift coefficients between the results of LES and the previous experiments, but experimental data are not indicated determinately and scattered. The increase in the standard deviation of drag and lift between $D/B = 2.0$ and 3.0 can be qualitatively simulated in this LES as shown in Figure 12(c) and (d).

The pressure distribution along the side surface of a rectangular cylinder is determined by the behaviors of separated shear layers, which are highly influenced by alternate vortex in wake. The pressure distributions along the side surface of $D/B = 3.0$ are shown in Figure 13. Time-averaged values are satisfied as a whole. While, there is considerable difference in the standard deviation of pressure between the experiments by Nishimura and Taniike and LES. But the position of increase in fluctuating pressure is in good agreement among LES and experiments. In case of $D/B = 3.0$, the separated shear layers from the leading edges start a partial reattachment on the side surface of the cylinder. The separated shear layers become
turbulent before natural development of separated shear layer flowing along the side surface. Consequently, as a reproduction of delicate turbulent transition not required, the pressure distributions can be predicted with a certain level of accuracy by LES.

(a) Non-uniform Cartesian grid and boundary-fitted grid  
(b) Grid close to the cylinder surface
Figure 3: Overset grid system for a circular cylinder.

(a) Central difference scheme  
(b) Upwind scheme
Figure 4: Vortical structures around a circular cylinder (Instantaneous vorticity contours).

Figure 5: Effect of the numerical dissipation on the mean pressures on a circular cylinder.

(a) Mean pressure coefficients  
(b) Fluctuating pressure coefficients
Figure 6: Effectiveness of a new pressure solver for pressures on a circular cylinder.
Figure 7: Overset grid for a circular cylinder. Figure 8: Instantaneous velocity contours.

Figure 9: Velocity vectors near separation point for a circular cylinder (Re=6*10^6).

Figure 10: Overset grid system for a rectangular cylinder with side ratio equal to 2.0.

Figure 11: Coordinate system for a rectangular cylinder.
Figure 12: Aerodynamic characteristics for a rectangular cylinder with various side ratios.

Figure 13: Pressure distributions on the side surface of a rectangular cylinder with $D/B = 3.0$. 
Aerodynamics of a three-dimensional square cylinder in turbulent boundary layer

This session deals with the aerodynamics of a three-dimensional square cylinder immersed in the turbulent boundary layer. This is a very simple situation where a cylinder with very fundamental shape of section such as a square is placed in isolation without any other surrounding buildings subject to the turbulent approaching flows. However fundamental issues such as turbulence effects, three-dimensional effects and unsteady flows with massive separation etc. are included in this example for discussing the potential of CFD to evaluate wind loads. After this session, all the issues dealt have a specified building inside a turbulent boundary layer, so the generation of inflow turbulence has been required. According to DNS results provided by Nozu and Tamura (2006), predictive accuracy of CFD for fluctuating characteristics of wind flows, pressures and forces on a three-dimensional square cylinder in turbulent boundary layer have been examined.

They employ the numerical method where the convection term is very accurately discretized by the fourth-order central finite difference scheme with fourth-order interpolation and no SGS model is incorporated. But for stable computation the upwind concept is implemented to the time-advancing integration.

Figure 14 illustrates the numerical model by the Cartesian grid system for a three-dimensional square cylinder in turbulent boundary layer consisting of the driver domain for inflow turbulence and the main computational domain for the cylinder flows. For driver domain, two types of turbulent boundary layers with the roughness exponent $\alpha_u=1/7$ and $1/4$ have been numerically simulated using the periodic boundary condition in the streamwise direction as same as a half channel turbulent flow simulation. For main computational domain, the nesting technique is applied for the adjacent region around a square cylinder. In nesting procedure, a finer grid system is gradually employed towards the target object.

Figure 15 depicts the numerical example obtained by the nesting technique. According to the instantaneous pressure contours around a three-dimensional square cylinder, the finer structures have been definitely recognized in the finer grid and varied in space smoothly. It can be confirmed that the transport of the computed data between each region works well.

Figure 16 shows the power spectra of wind velocity fluctuations at the cylinder height and fluctuating lift for a three-dimensional square cylinder in turbulent boundary layer. Both normalized spectra of velocity fluctuations for uniform and $1/4$ shear flows are in reasonably good agreement with the experimental data except the high frequency region. Reductions of the computed data in the higher frequency can be considered due to the grid resolution and the Reynolds number. Turbulent integral scale of the computed turbulence can be estimated approximately twice cylinder breadth, which is almost 60% of the scale in experiment. Concerning the spectra of fluctuating lift, the experimental and numerical data are coincident with each other. The case of uniform flow show the sharp peak corresponding to vortex shedding frequency, but its power is much smaller than the case of $1/4$ shear flow. The shear flow case has the spectra of larger power with a broad band.

Figure 17 presents the pressure distributions in the horizontal section for a three-dimensional square cylinder in turbulent boundary layer. Consistency with experimental data is very well for all the data of time-averaged, fluctuating and peak coefficients. Only for the fluctuating and the peak pressure coefficient at the side just behind the separation point, small differences can be seen and both indicate a smaller value compared to the experimental data. This is due to the insufficient power of incident flows in relatively low frequency region.

Figure 18 shows the spectra of pressure fluctuations on the surface of a square cylinder. Frontal, side and back surfaces have respectively their own characteristics related
with the turbulence of incident flow or vortex-shedding effects. It can be recognized that these characteristics have been simulated with sufficient accuracy by CFD for the experimental data.

(a) Driver domain (top: \(\alpha_u = 1/7\) shear flow, bottom: \(\alpha_u = 1/4\))  
(b) Main domain

Figure 14: Numerical model for a 3D square cylinder in turbulent boundary layer.

Figure 15: Numerical example obtained by the nesting technique (Instantaneous pressure contours around a three-dimensional square cylinder).

(a) Wind velocity fluctuations at the cylinder height  
(b) Fluctuating lift

Figure 16: Power spectra for a three-dimensional square cylinder in turbulent boundary layer.
Urban canopy flows in the center of a city

Nozawa et al. (2009) have performed LES of complicated turbulent flows within and above the urban canopy. In order to establish a sophisticated numerical model for wind turbulence under urban environment, this study investigates the technological potential and the future trend of LES by employing a numerical model which directly represents an actual and complex shape of city by urban composite elements, such as buildings, structures, vegetation, or various kinds of terrain. Also, detailed comparison of LES results with wind
tunnel experimental data has been carried out for the model validation. Although a city has so complicated shape, urban wind flows have been thus far investigated mainly by using homogeneously-arrayed roughness blocks on the ground. Alternatively, some canopy models have been proposed by many researchers, but their efficiency is ambiguous for the actual urban area. They focus inside the urban canopy where near-ground flows are very complex and unsteady due to separation and vortex shedding among building elements. A necessity of the model reproducing the near-ground feature is discussed.

Numerical model for representing the bottom surface configuration

The numerical model is consisting of the driver domain for inflow turbulence and the main region for urban wind flow, as shown in previous session. To simulate a rough surface in the driver domain, rectangular blocks are directly arranged on the surface. So, no assumption such as roughness length is used for the bottom boundary. External forcing method proposed by Goldstein et al. (1993) is employed to impose the no-slip boundary condition at all surfaces of a roughness element. In this IBM, the forcing term is added to the governing momentum equations where the coupling of the external force and the time derivative acceleration term corresponds to the equation for a kind of oscillation system with an over-damping.

For the main domain, the aspect of configuration at the near-ground surface in the actual urban area can be also treated in same manner, so the boundary condition at the stepped surface by a building and the ground level is imposed by the above forcing technique. The height data for buildings are derived from 2m resolution GIS data (RAMS-e, KOKUSAI KOGYO CO.).

Experimental setup

Wind tunnel test is carried out at Building Research Institute to verify the accuracy of LES comparing the mean velocity profiles in urban area. Test section of the wind tunnel is 3.0m width and 2.5m height (Figure 19) and the model scale is set to be 1/750. This specified urban area has two areas of Marunouchi and Yaesu separated by the Tokyo Railroad Station. Wind directions are from south-southwest. For the incident flow, the vertical profile of streamwise mean velocity fits to exponential law with its exponents $\alpha_u=0.33$. Mean velocity vertical profiles along the street canyon are measured using a thermistor anemometer.

Generation of inflow turbulence

The quasi-periodic boundary condition is introduced in streamwise direction to simulate the spatially developing boundary layer for generating turbulent inflow. The quasi-periodic boundary condition proposed by Nozawa and Tamura (2001) has been applied to a rough-wall turbulent boundary layer flow. In this method the velocities at the recycle station are rescaled and reintroduced at the inlet. For roughness parameter used at rescaling, the resistance formula of sand-roughened plate by Prandtl (Schlichting, 1979) is employed.

Figure 20 shows mean velocity profiles generated using the quasi-periodic boundary condition. The computational domain of generating inflow is 2.5km length and 2km width. In order to simulate the mean velocity profile of experiment, roughness density of simulation is set to 7.5%. Agreement of both data with each other can be surely realized. The boundary layer has developed to almost 1km of thickness for 2km height of a computational region.

Numerical results

Figure 21 illustrates the main computational region horizontally with 2.5km length in streamwise direction and 2km width in transverse direction. The horizontal mesh size is 4m in the both directions and vertically the lowest mesh is around 1m. Numbers of grid points are 625 in streamwise direction, 501 in transverse direction and 110 in vertical direction.
Figure 22 shows the wind speed correlation between experimental and LES data. High correlation of LES with experiment is obtained as such that the correlation factor is 0.991. From the results, it could be said that if turbulence inflow is appropriately generated LES could predict the wind speed profile in a city with sufficient accuracy not only above the canopy but also within the canopy.

Next, the difference of wind characteristics at each area is discussed. The Marunouchi area possesses many tall buildings, while the Yaesu area is occupied by low-rise and middle-rise buildings. The turbulence characteristics must be different between the two areas within the canopy since the size and the space of the roughness elements are quite different. Influence of this difference may range in whole region of a roughness sublayer. For comparison, turbulent statistics to spatial variation of mean velocity are obtained by averaging over 720m x 340m area for Marunouchi and 560m x 400m area for Yaesu (Figure 21). Mean height of buildings for Marunouchi is 65m and height of the tallest building is 197m. While, Yaesu has 27m as mean height and 75m as the tallest height. Figure 23 shows the profiles of volume density for buildings. This density in Yaesu area rapidly decreases upward, although that in Marunouchi decreases gradually. These decreasing lines for Marunouchi and Yaesu have crossed at 25m high.

Figure 23 shows the mean velocity profiles at various points for both Marunouchi and Yaesu areas. Mean velocity profiles collapse above 267m for Marunouchi and above 100m for Yaesu. The collapsing heights are almost 4 times as high as the mean height for both areas. The spatially-averaged Reynolds shear stress profiles and dispersive shear stress profiles are compared for various type of urban roughness in Figure 24. Here dispersive shear stress is defined as follows.

\[
\langle u^*w^* \rangle = -1/A \int \left[ (\bar{u} - \langle \bar{u} \rangle) (\bar{w} - \langle \bar{w} \rangle) \right] dA
\]

Dispersive shear stress represents a contribution to momentum transfer from correlations between point-to-point variations in the time-averaged flow.

Strong shear of mean velocity profile in the wake of a roughness element increases the spatially averaged Reynolds shear stress, besides increases the turbulent energy. On the other hand, the dispersive shear stress increases the turbulent energy directly (wake production). At the inlet, spatially-averaged Reynolds shear stress profile has a local peak at 85m high. This height almost corresponds to the height (75m) of a roughness block in the driver domain. At Yaesu, spatially-averaged Reynolds shear stress profile almost corresponds to that of the inlet, while the peak height comes down to almost 65m high which is twice higher than the mean height at Yaesu. The mean wind speed profiles have strong shear mainly at range from 27m to 53m high. Because of this strong shear the spatially-averaged Reynolds shear stress increases at lower area than the peak height of the inlet. While, Marunouchi has strong shear at range from 133m to 150m and from 200m to 267m and this might contribute to exaggerate the broad peak (from 140m to 210m high) in the spatially-averaged Reynolds shear stress profile.

Dispersive shear stress at Yaesu has a peak at 50m height and it gradually decreases to be 10% of the peak value at 140m height. Estimating the upper limit of the roughness sublayer from the dispersive shear stress, it could be said that the upper limit corresponds to almost $5h$ ($h$: 27m, the mean building height). 90% of buildings in the Yaesu area are lower than 40m, as shown in Figure 23. So, in the case of this 40m as $h$, the upper limit of the roughness sublayer becomes almost $3.5h$. In the past study, the upper limit of the roughness sublayer formed in turbulence boundary layer flows over homogenous roughness is estimated 2-5$h$ and these estimations at Yaesu are in this range in spite of actual urban area. Dispersive shear stress at Marunouchi has a peak at the height of the tallest building (175m high) and
declines just above this height. This stress does not decrease to 10% of the peak value even at $5h$ ($h$: 65m). There are several local peaks at the range from 26m to 102m high. These local peaks are strongly related to each height of various buildings because the buildings at Marunouchi are built much sparser than those at Yaesu. The boundary layer formed at Marunouchi may not fully develop because of a short fetch. Then it is difficult there to estimate the upper limit of a roughness sublayer.

Figure 19: Photograph of wind tunnel test model for Marunouchi and Yaesu areas.

Figure 20: Mean velocity profiles at inlet for both simulations and experiments.

Figure 21: Computational region and measurement points (Wind direction: SSW).
Figure 22: Correlation of wind speeds between experimental data and LES data.

Figure 23: Volume density of buildings and mean wind speed profiles at each area. (left: volume density, center: Marunouchi, right: Yaesu).

Figure 24: Profiles of spatially-averaged Reynolds shear stress and dispersive shear stress.
Wind flows around a tall building in actual urban area

This session has aimed at estimating wind velocity and wind turbulence in an actual urban area by combination of LES and MM5 techniques. Kishida, Tamura and Takei et al. (2008, 2009) have investigated the effectiveness of this combined model for wind prediction in a city. As a meteorological event, they consider the high wind during cyclogenesis generated near Japan. They deal with two examples for the urban area in the center of Tokyo, one is around an isolated tall building and the other is an urban canopy area formed by multiple tall buildings. A numerical model in an actual urban area has been constructed using GIS data with high resolution, and the inflow boundary condition has been appropriately given by numerically simulating the turbulence characteristics of natural wind flow over the urban area. Also, this study performs the detailed comparison of LES results with full-scale measurement for the model validation. However LES gives only a value relative to the reference wind speed. In order to introduce the absolute value, the mesoscale meteorological model (for example, 5th Generation Mesoscale Model: MM5) is utilized here. That is to say, they showed an evaluation of the absolute values in LES by combining MM5 and LES results at the reference location, such as above the urban canopy (Tamura and Kishida et al., 2009, Kishida and Tamura et al., 2009), above the roughness sub-layer or the internal boundary-layer height (Tamura and Takei et al., 2008). Accordingly, they can also set up the approaching wind turbulence with evaluating the specific level of wind speed over the actual urban area.

Comparison of MM5 results with observational data

Figure 25 shows the wind velocity at a higher point of specific locations in Tokyo obtained by MM5 with the observational data at the top of several high-rise buildings (Tamura and Kishida et al., 2009, Kishida and Tamura et al., 2009, Tamura and Takei et al., 2008). This study deals with observational data during cyclogenesis passed near Japan in January, 2006. On the whole, the wind directions estimated by MM5 are surely in good agreement with the observational data. For Roppongi, the wind speed estimated by MM5 is almost in good agreement with the observational data. This area has only an isolated high-rise building and is otherwise uniformly covered by medium-rise buildings. The MM5 results at Roppongi are hardly influenced by the urban canopy layer. However, for Ootemachi and Marunouchi, the wind speeds estimated by MM5 are larger than those of the observational data because of the lower measurement height and the existence of surrounding buildings. It can be considered that the observational results are greatly influenced by the urban canopy layer, so the wind speed by MM5 was greatly overestimated. In the case of the area with densely-arrayed high-rise buildings, the meteorological model has a limitation for the prediction of the wind flows within the urban canopy. So we can say that LES technique is expected for estimation of the local canopy flows.

Wind flows around an isolated tall building

Figure 26 illustrates a numerical model for LES at local urban area in Roppongi of Tokyo. Here, the numerical model consists of two computational domains, one is for the main domain of turbulent flow over actual urban area, and the other is a driver region for the auxiliary simulation for generating inflow turbulence for the main domain. To generate inflow turbulence, the modified method by Nozawa and Tamura (2002) is employed for a rough-wall turbulent boundary layer. Table 1 shows the size and the grid numbers of the computational domains for the case of an isolated tall building.

Generally, LES cannot evaluate the absolute wind speed but only a value relative to the reference wind speed. The above MM5 results are used to introduce the absolute wind
speed for LES data. In order to combine LES data with MM5 data, the reference point for wind must be given in the computational domain. Considering that the observational data at Roppongi show sufficiently good agreement with MM5 results, a reference point has been determined 750m upwind of the rooftop of the high-rise building at a height of about 250m. Accordingly, the LES wind velocity at the reference point can be given as a dimensional value by using the predicted absolute value of MM5 at the same height.

Figure 27 illustrates the instantaneous wind flow patterns around an isolated high-rise building. We can recognize the turbulent flow structures with various scaled vortices in the whole computational domain, because turbulent boundary layer is imposed at inflow condition. At a height of 100m, we can find a separated shear layer and vortex-shedding around the high-rise building. Near the ground, dominant flows with high velocity can be seen along the main street.

Figure 28 compares the LES results with the observational data for the time series of wind speed at the rooftop over Roppongi area during ten minutes. We can see that the time history at the rooftop is slightly affected by a building obstacle and shows a little larger variation. We can also see a low-frequency fluctuation in the observational data. Mean wind speed, gust factor and turbulence intensity in the table estimated by LES and MM5 are in relatively good agreement with the observational data.

Figure 29 illustrates the instantaneous wind speed vectors and contours in the near wake of the high rise building. It can be seen that the wind speed at point 4 becomes high because the flows converged inside the street near the high rise building. The position of the point 6 is behind the high rise building. The reverse flow with fine structures can be recognized, fluctuation of wind direction is presumed to be very large in this region.

Figure 30 illustrates a numerical model for LES at the Marunouchi area in Tokyo. We can recognize several tall buildings at computational main regions and imagine the urban canopy layer is formed at this area. Table 2 shows the size and the grid numbers of the computational domains for urban canopy.

Figure 31 illustrates the instantaneous wind flow patterns within an urban canopy at Marunouchi area of Tokyo. We can recognize the turbulent flow structures with various scaled vortices among complex buildings, because turbulent boundary layer is largely deformed by very rough condition at the ground surface.

The LES wind velocity at the reference point can be given as a dimensional value by using the absolute value predicted by MM5 at the same location. Here, they consider two cases of height positions as a reference point and examine applicability of a reference velocity at the reference point to transform the LES results to be dimensionalized. One case is set to the height of 125m (about five times of averaged building height) and the other is 250m (about ten times of averaged building height).

Table 3 compares the wind velocity at the rooftop among observational data, MM5 results and LES results based on MM5. The LES+MM5 results are closer to the observational data than the only MM5 data. According to Figure 31, the height of 125m is within the urban canopy even at five times height of averaged building height. So the case predicted by the reference height of 250m show the better results for the wind velocity at the rooftop.

Figure 32 compares the LES results with the observational data for the time series and spectra of wind speed at the rooftop height over Marunouchi area for ten minutes. We can recognize that the time histories of both LES and observation at the rooftop are much affected by the surrounding tall building and show a much larger fluctuation. But the spectra are almost the same in equilibrium state of turbulence and the rooftop data tend to hardly have a very small shift to high-frequency region by the surrounding circumstances. Gust factor and
turbulence intensity in Table 4 estimated by LES are in satisfactorily good agreement with the observational results.

Figure 25: Comparison between MM5 results and the observational data on the rooftop of high-rise building at the center of Tokyo

Figure 26: Numerical model for an isolated tall building

Table 1: Computational region for an isolated tall building

<table>
<thead>
<tr>
<th>Driver region</th>
<th>Main region</th>
</tr>
</thead>
<tbody>
<tr>
<td>X</td>
<td>Y</td>
</tr>
<tr>
<td>Domain(km)</td>
<td>2.5</td>
</tr>
<tr>
<td>Grid points</td>
<td>500</td>
</tr>
<tr>
<td>Grid size(m)</td>
<td>5</td>
</tr>
</tbody>
</table>

Figure 27: Instantaneous wind velocity around a high-rise building

Figure 28: Comparison of LES and observation; time-series of wind velocity at higher position

Figure 29: Instantaneous wind velocity vectors and contours

Figure 25: (a) Roppongi (b) Otemachi (c) Marunouchi

Figure 26: (a) Driver domain (b) Main domain

Figure 27: (a) Upper wind (at 100m high) (b) Near the ground surface

Figure 28: (a) Observation data MM5(250 meters high) (b) Observation data MM5(75 meters high) (c) Observation data MM5(180 meters high)
Table 2: Computational region for urban canopy

<table>
<thead>
<tr>
<th>Driver region</th>
<th>Main region</th>
</tr>
</thead>
<tbody>
<tr>
<td>X</td>
<td>Y</td>
</tr>
<tr>
<td>Domain(km)</td>
<td>2.4</td>
</tr>
<tr>
<td>Grid points</td>
<td>600</td>
</tr>
<tr>
<td>Grid size(m)</td>
<td>4</td>
</tr>
</tbody>
</table>

Table 3: Comparison among observational data, MM5 results and LES+MM5

<table>
<thead>
<tr>
<th>Relevant building</th>
<th>Meteorological observatory</th>
</tr>
</thead>
<tbody>
<tr>
<td>Observational data</td>
<td>10.0(m/s)</td>
</tr>
<tr>
<td>MM5 results</td>
<td>16.6(m/s)</td>
</tr>
<tr>
<td>LES+MM5(combination height:125m)</td>
<td>14.1(m/s)</td>
</tr>
<tr>
<td>LES+MM5(combination height:250m)</td>
<td>11.9(m/s)</td>
</tr>
</tbody>
</table>

Table 4: The comparison of LES results and observational data at rooftop height

<table>
<thead>
<tr>
<th>Gust factor</th>
<th>Turbulent Intensity</th>
</tr>
</thead>
<tbody>
<tr>
<td>Observational data</td>
<td>1.48</td>
</tr>
<tr>
<td>LES(building top)</td>
<td>1.52</td>
</tr>
<tr>
<td>LES(upstream)</td>
<td>1.30</td>
</tr>
</tbody>
</table>

Figure 30: Numerical model for urban canopy

Figure 31: Instantaneous wind velocity in urban canopy

Figure 32: Comparison of LES and observation for wind velocity at rooftop height.
(Left: Time series, center: Spectra of the LES data, right: Spectra of the observation data)
Wind pressures acting on a tall building in urban area

This final session discusses the predicted values of wind pressures acting on a tall building which is surrounded by other tall buildings in urban area. This area is at the center of Tokyo and a very deep urban canopy layer has been definitely formed around a specified tall building. Kishida and Tamura et al. (2009) have performed the estimation of wind velocity and wind turbulence in an actual urban area by combination of LES and MM5 techniques. As the meteorological event, they consider the typhoon 0416 which passed through the north-west side of Japan on 30th-31st August, 2004. In this case, south wind was blowing in the Tokyo area more frequently during the period with a high wind velocity. For rooftop data at 250m, the wind velocity estimated using the MM5 is almost in good agreement with the observational data. It can be confirmed that MM5 data have been used for getting the reference velocity. Based on these results, they have presented LES results of the aerodynamic characteristics for the specified tall building as well.

Figure 33 depicts the LES model for a tall building in dense city district. We can recognize that the specified building faces a few tall buildings in the upwind direction. One of them is very close to the specified building and has a possibility to affect the formation of a wake or vortices. A distant building may also generate a high wind by the sheltering effect. Then a computational example has been added to elucidate the effect of surrounding circumstances. A numerical model without the surrounding other buildings is made by giving the limitation of 45m for the building height.

So, after checking the wind velocity data at roof top of the specified tall building by the coupled method of LES and MM5, they have moved to the analysis on details of wind flows inside the urban canopy and the pressures acting on the tall building.

Figure 34 illustrates the instantaneous wind flow in the urban canopy represented by the velocity vectors and velocity contours. We can see the distant tall building has generated the high wind which is attacking to the specified tall building. Near tall building cannot make a wake freely and as a result the wake structures are much deformed. On the other hand, According to instantaneous wind velocity and pressure around the specified building in Figure 35, the vortex shedding from the specified building is not symmetric due to the near building. Also, we can recognize very high wind due to a kind of valley flow between the adjacent building and the specified building.

Figure 36 present the mean pressure distributions acting on a surface of the specified building. In the case without the surrounding other buildings, the vortex shedding is symmetrical, so the pressure distribution shows the values consistent with the previous experimental data. Large negative pressures appear at the upwind location of the side surface where the high wind occurs due to the effect of the adjacent building. As a result of the bias of vortex shedding, the stagnation point with highest positive pressure on the frontal surface moves to the opposite location of the adjacent building. They try to compare the LES data with previous experimental data (Figure 37), but the circumstance of building array easily has changed year by year because a new tall building is actively built at the center of a big city. The complete comparison in the urban area is extremely difficult. Understanding these situation, the comparison with old-time experimental data indicates the LES data for recent circumstance are much more affected by the surrounding other buildings and the old-time experimental data fit to the values between the present LES data and the data obtained by the case without the surrounding other buildings.
Figure 33: Numerical model for a tall building in dense city district.

Figure 34: Instantaneous wind velocity in urban area (velocity vectors and velocity contours).

Figure 35: Instantaneous wind velocity and pressure around the specified building. (velocity vectors and pressure contours)
Figure 36: Mean pressure distributions acting on the specified building.
(left: frontal and back surfaces, right: side surfaces)

Figure 37: Experimental data of mean pressure distributions on the specified building

Conclusions
This paper is reviewing the researches with regard to the large eddy simulations on estimation of wind effects on buildings and structures.
First, applicability and effectiveness of various numerical techniques used for the LES model of turbulent wind have been elucidated by surveying recent researches. Second, turbulent flows and pressure distributions around a bluff cylinder with a simple sectional shape are given as fundamental numerical examples and investigated in order to evaluate a current stage of LES for the physical mechanism to be understood. Third, with regard to a tall building with actual shape existing among densely arrayed other tall buildings in a city, wind flows and aerodynamic characteristics for practical use have been examined by the sophisticated model based on both of LES and meso-meteorological model, furthermore its predictive accuracy has been discussed in comparison with the field measurement data as well as wind tunnel experimental data.

The obtained results are summarized as follows:

1. The SGS models proposed recently are expected because the model generally becomes a friendly use. The overset grid method is a very powerful tool to get the accurate results for the flows around a bluff cylinder. The artificial diffusion has a large effect on the vortex formations and the following aerodynamics, especially a fluctuating part. Inflow turbulence has a dominant role to determine the aerodynamic characteristics.

2. For the fundamental issues of a bluff cylinder with simple section, LES can give a result with sufficient accuracy with regard to the super critical Reynolds number flows, the reattaching flows and its interaction with the singular point at the corner and the spectra as well as the peak values for fluctuating physical quantities.

3. For the application issues of urban canopy flows, GIS data can be used to construct the numerical model correctly reproducing the near-ground configuration. Features consisting of tall or large buildings and main streets have a large effect on unsteadiness and complexity in the formation process of flow structures inside the canopy. The present LES can predict mean wind profiles within and above urban canopy with very good accuracy.

4. For the application issues of wind flows around a tall building in urban area, the LES model coupled with the meteorological model is useful. The applicability and effectiveness of this hybrid approach is clarified to predict strong winds blowing among the densely arrayed tall buildings. The predictive values obtained from the coupling of the MM5 and the LES corresponds quite well to the observation data not only for the mean wind velocity but also for fluctuating part such as the gust factor. Also, it can be confirmed that the model can appropriately determine the height of a reference point for combining both the data.

5. For the application issues of wind pressures and wind forces acting on a tall building in urban area, the neighboring circumstances as well as the specified tall building should be reproduced correctly. The features of the elevated buildings in urban canopy sensitively affect the flow patterns such as vortices, a separated shear layer and horse-shoe vortex, which determine the aerodynamic characteristics.

6. This reviewing has elucidated that we are now standing at the stage to employ LES technique for solving the unresolved practical problems in the designing process of wind engineering. Especially, we have a potential to evaluate the wind resistant performance of buildings and structures with sufficient accuracy. So, an example of CFD application in structural design under the strong wind will appear soon.
References


