

Inflow turbulence generation for LES of wind flow around bluff body

B.W. Yan^a, Q.S. Li^a

^a *Department of Civil and Architectural Engineering, City University of Hong Kong, Hong Kong*

ABSTRACT: Large eddy simulation (LES) has been widely applied in wind-related problems. A key issue to ensure obtaining accurate results by LES is to generate an appropriate inflow field satisfying the physical wind characteristics in the atmospheric boundary layer (ABL). The methods adopted in this study for inflow field generation include two statistical methods (the random flow generation method (RFG) and the synchronizing random flow generation method (DSRFG)) and a numerical method (the recycling method). These inflow generation methods are used to predict the wind effects on the Commonwealth Advisory Aeronautical Council (CAARC) standard tall building. The computational results show that all the three methods can reproduce the time-averaged velocity and turbulence intensity profiles quite well at the building position. However, the RFG method is incapable of generating fluctuating velocity satisfying the prescribed von Karman spectrum, and the DSRFG method yields random flow field satisfying the target spectrum at the inlet, while the recycling method generates the streamwise velocity fluctuation quite well at the inlet. The mean pressure coefficients on the front surface of the building are accurately predicted by all the inflow turbulence generators (the three methods), and the pressure coefficients distribution patterns on the side and back surfaces are well predicated, while some discrepancies still exist between the numerical results and wind tunnel measurements. All the RMS pressure coefficients obtained by the inflow turbulence generation methods fall in the range of wind tunnel data, illustrating that the effects of turbulence characteristics on the wind-induced pressures on the tall building are reproduced to some extent.

KEYWORDS: inflow turbulence generation; CFD, LES; bluff body; wind effect

1 INTRODUCTION

Large eddy simulation (LES) has been developed since the 1960s. But, its application to industrially relevant problems has only recently become available as LES requires sufficiently powerful computing resources. With developments of computational fluid dynamics (CFD) techniques, LES has been widely applied to simulate turbulent flows of engineering interest. In computational wind engineering (CWE), LES has become a powerful tool which has been applied in the predication of wind effects on buildings and structures, wind flow characteristics and topographical effects, wind fields in densely-arranged cities and atmospheric environments [1]. For the application of LES in atmospheric boundary layer, a very important issue to enable the LES techniques to obtain accurate results is to generate a physically appropriate flow field as an inflow boundary condition (inflow turbulence) satisfying prescribed spatial and temporal wind characteristics, since inaccurate generations of inflow turbulence can result in inaccurate simulation results of wind effects on buildings and structures. Several inflow turbulence generation techniques have been proposed in the past decade. They can be classified into three categories: synthetic method, precursor simulation, and recycling method.

The simplest method is to superimpose the random noise with proper turbulence intensity on the measured mean velocity distribution. But, such a method requires sufficient field measurement information from ground to the gradient height, which is usually not available, and this method ignores the cross-correlations between velocity components, namely Reynolds stresses

and spatial relations in turbulence structures. And the Lack of the spatial information can be solved by taking considerable settling distance to simulate the entire upstream region before the mean flow reaches the equilibrium boundary layer. Instead of using the problem-dependent settling distance, the recycling method was introduced by Spalart [2] using the periodic boundaries to reduce the settling distance. This method was further developed by Lund et al. [3] who introduced the rescaling idea to specify a precise inlet profile satisfying the prescribed shear stress and momentum thickness. Kataoka and Mizuno [4] simplified Lund's rescaling method using the constant boundary layer thickness in place of the similarity laws of the boundary law. Nozawa and Tamura [5] further developed Lund's method for generation of boundary layer conditions. Their method is similar to the method employed in wind tunnel tests to simulate boundary layer conditions. Liu and Pletcher [6] indicated that the position of the recycling and rescaled plane could cause a difficulty resulting from the coupling of the initial and inflow condition which will influence the computational cost, and they proposed a modification to Lund's method by using dynamically positioned recycling plane according to instantaneous flow properties. All the above mentioned methods demand an auxiliary LES computation with streamwise periodic boundaries to specify a fully developed inlet profile for the flow around complex geometries as the starting point. For the synthetic methods, one common way is to obtain the velocity fluctuations from an inverse Fourier transform of prescribed spectrum which follows the prescribed power spectrum density and spatial correlations. This method was initiated by Hoshiya [7], then developed by Iwatani [8], and Kondo et al. [9]. Another synthetic approach is based on proper orthogonal decomposition (POD) and linear stochastic estimation (LSE) in combination with the simultaneously measured fluctuating velocity obtained by hot wire probes or stereoscopic particle image velocimetry (SPIV) conducted by Perret et al. [10]. The methods based on the variant of spectral method for generation of an isotropic continuous flow field were further developed by several researchers including Smirnov et al. [11], Klein et al. [12], Batten et al. [13] and Huang et al. [14]. Smirnov et al. [11] superimposed turbulence length scale and time scale into the basic model proposed by Kraichnan [15] to generate an isotropic divergence-free fluctuating velocity field with the target length scale and time scale, and they realized the inhomogeneous and anisotropic turbulence characteristics through a scaling and orthogonal transformation of the resulted flow field with a given anisotropic velocity correlation tensor. Huang et al. [14] further developed the Kraichnan's approach to establish a general inflow turbulence generation method capable of generating a spatially correlated turbulent flow field satisfying any target spectrum including the von Karman spectrum through the discretizing and synthesizing procedures. Another method, the precursor simulation, was conducted to run a separate precursor calculation of an equilibrium flow to generate a library of turbulent database that can be reintroduced into the main simulation. Although the method demands extra computational efforts and cost to establish the database, the inflow field should possess the main characteristics of turbulence structures including temporal and spatial fluctuations and a reasonable energy spectrum.

2 INFLOW TURBULENCE GENERATION METHODS

The methods adopted in this study are the random flow generation method by Smirnov et al. [10] and the discretizing and synthesizing random flow generation method by Huang et al. [14], and the recycling and rescaling method based on Kataoka's research works [4].

2.1 Random flow generation method (RFG)

The spectral synthesizer in the Fluent code is based on the RFG method modified by Smirnov et al. [11]. The method can be used to generate fluctuating velocities which are computed by synthesizing a divergence-free velocity vector field from the summation of Fourier harmonics on the basis of the input turbulence boundary conditions. In the implementation of the Fluent code, the number of the Fourier harmonics is fixed to 100 [16].

2.2 Discretizing and synthesizing random flow generation (DSRFG)

Another approach employed in the study is DSRFG, which is based on Kraichnan's method to establish a general inflow turbulence generator capable of generating a spatially correlated turbulent flow field satisfying any target spectrum including the von Karman spectrum. A brief formulation of the method is presented below [14].

$$u(x, t) = \sum_{m=k_0}^{k_{\max}} u_m(x, t) = \sum_{m=k_0}^{k_{\max}} \sum_{n=1}^N \left[p^{m,n} \cos(\tilde{k}^{m,n} \cdot \tilde{x} + \omega_{m,n} t) + q^{m,n} \sin(\tilde{k}^{m,n} \cdot \tilde{x} + \omega_{m,n} t) \right] \quad (1)$$

$$\text{where } p^{m,n} = \frac{\zeta \times k^{m,n}}{|\zeta \times k^{m,n}|} \sqrt{a \frac{4E(k_m)}{N}}, q^{m,n} = \frac{\xi \times k^{m,n}}{|\xi \times k^{m,n}|} \sqrt{(1-a) \frac{4E(k_m)}{N}}, \tilde{x} = \frac{x}{L_s}, \tilde{k}^{m,n} = \frac{k^{m,n}}{k_0}, |k^{m,n}| = k_m,$$

$\omega_{m,n} \in N(0, 2\pi f_m)$, $f_m = k_m \times U_{\text{avg}}$, ζ and ξ are the vector form of ζ_i^n and ξ_i^n . L_s is the turbulence integral length scale which denotes the two-point spatial correlation. a is a random number uniformly distributed between 0 and 1. U_{avg} is the average velocity at the reference height. $N=100$ (sampling number for each wave number k_m).

The artificial velocity fluctuations generated by the turbulence generation method will be resolved by the grid which is commonly chosen to be the spatial filter in LES. This means that the artificial velocity will be transformed into velocities and SGS viscosities through the chosen SGS model. For this reason, the grid scale between the main building and the inlet of the computational domain should be fine enough to enable most of the artificial velocity fluctuations generated at the inlet can be resolved and then acting on buildings.

2.3 Recycling method

The recycling and rescaling method is based on the approach proposed by Kataoka and Mizuno [4], which is a simplified version of Lund's method. In the modified method, the development of the boundary layer thickness is neglected and treated as constant and the mean velocity profile is fixed at inflow. As shown in Figure.1, the approach is to extract instantaneous velocity fluctuations from the recycled station and reintroduce the rescaled fluctuations at the inlet in an auxiliary simulation of a zero pressure gradient boundary layer, which is denoted as driver section. The auxiliary simulation is spatially developing through the rough wall surface, and generates its own inflow conditions which make the approach convenient for an existing inflow–outflow computational domain that can be converted to the inflow turbulence generator after a sequence of simple operations. The coupled inlet and recycling plane are defined as “pseudo-periodic condition”, and the periodic boundary conditions in the streamwise direction are appropriate for fully developed flow without a need for a development section. In this study, the inflow generation calculation in the driver region is synchronized with the main simulation in the main domain (shown in Figure.2). The velocity fluctuation was generated based on Kataoka's method [4], and the rough wall surface is realized with the cubic roughness elements arranged in the streamwise direction.

L is the width of the cubic roughness element while G is the center distance between the neighboring roughness elements. In this case, G is chosen as $5L$, and such an arrangement gave a roughness density $\lambda=8.0\%$. In addition, the generation of the Gaussian random perturbations is superimposed in the top region at the inlet for the recycling method in the auxiliary computation since the lower turbulence intensity generated by the recycling method was observed in recent study. The velocity components prescribed at the inlet following Kataoka's method [4] are given as follows:

$$\begin{aligned} u_{inlt(y,z,t)} &= \langle u \rangle_{inlt}(z) + \phi(\theta) \times \{u(y,z,t) - \langle u \rangle(y,z)\}_{recy} \\ v_{inlt(y,z,t)} &= \phi(\theta) \times \{v(y,z,t) - \langle v \rangle(y,z)\}_{recy} \\ w_{inlt(y,z,t)} &= \phi(\theta) \times \{w(y,z,t) - \langle w \rangle(y,z)\}_{recy} \end{aligned} \quad (2)$$

The following damping function [16] was adopted to restrict the velocity fluctuations from developing for the unstable conditions.

$$\phi(\theta) = 0.5 \left\{ 1 - \frac{\tanh \left[\frac{8.0(\theta - 1.0)}{-4.0(\theta - 0.3) + 0.7} \right]}{\tanh(8.0)} \right\} \quad (3)$$

where $\theta = z / \delta$, δ is the boundary layer thickness.

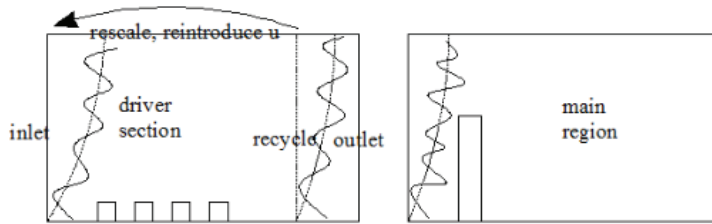


Figure.1 Schematic view of the recycling method

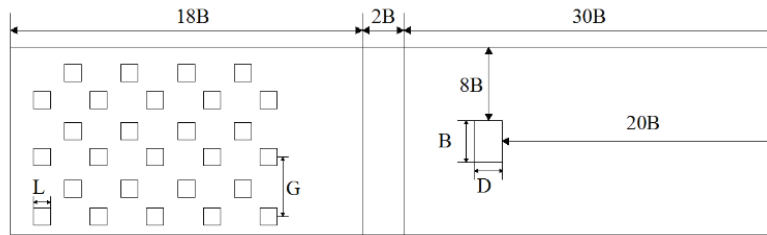


Figure.2 Roughness arrangement and dimensions of the computational domain

3 BUILDING MODEL, COMPUTATIONAL DOMAIN, MESH ARRANGEMENT AND BOUNDARY CONDITIONS

3.1 Building model

In this study, the selected building is the Commonwealth Advisory Aeronautical Council (CAARC) standard tall building. The building shape is rectangular prismatic and its full-scale dimensions are: sides 30.8 m by 45.72 m and height 183.88 m with density 160 kg/m^3 and natural frequency 0.2 Hz. As introduced by Melbourne [18], the building is flat-topped, without parapets, and the external surfaces are flat without mullions or other geometric disturbances. Since

the 1970s, the building model has been intensively investigated in various wind tunnels as a standard model to study the wind effects on tall buildings and several wind tunnel measurements about this building model were reported [19,20].

3.2 Computational domain

A 1:250 scale rigid model of the CAARC building is considered in this study. The main computational domain is shown in Fig.2. The Reynolds numbers involved in the simulations is the same as that in the wind tunnel experiment conducted by Obasaju [19] in British Maritime Technology (BMT) Environmental Wind Tunnel. As shown in Fig.2, the computational domain covers $30B$ (B is the breadth of the building model) in the streamwise direction, $17B$ in the spanwise direction and $2H$ (H is the height of the building model) in the vertical direction. The blockage ratio of the computational domain is nearly 3% which is regarded as acceptable in CWE community. In consideration of the periodic boundary in the spanwise direction, the size of the across-wind dimension is chosen to be at least twice the size of the largest eddy.

3.3 Mesh arrangement

For the auxiliary driver section using the recycling method, structured mesh arrangement is applied to discretize driver section, and for the need of capturing the turbulence generated by the roughness elements, the grid points are clustered near the cubic element surfaces and the ground neighborhood. About 1.2×10^6 mesh elements are used in discretizing driver section.

For the mesh arrangements in the main region, the structured mesh is used in the domain far from the building model and the unstructured mesh is adopted near the building model surfaces, since the Fluent code [16] is capable of dealing with both structured and unstructured grids. The advantages of this kind of mesh arrangement are capable of refining the grid distribution in the building surface neighborhood and avoiding excessive grid-stretching and reduce the total grid number in the main region. On the surfaces of the building model, a boundary layer with 20 grids is adopted and the first grid height next to the building surface is chosen as $B/1000$ and a growth factor 1.05 so that the near-wall grid y^+ ($y^+ = \rho u y / \mu$) is in the order of unit. Finally, the total grid number in the main region is almost 1.5×10^6 .

3.4 Boundary conditions

In order to obtain better agreement between wind tunnel experiment and numerical results, the inflow boundary conditions adopted in this study is identical as that in the wind tunnel experiment of Obasaju [19]. The log law wind speed profile of the atmospheric boundary layer simulated in the wind tunnel study of Obasaju [19] is given as follows:

$$\frac{U}{U_\tau} = 2.5 \ln\left(\frac{Z - D}{Z_0}\right) \quad (4)$$

where U_τ is friction velocity, D is the zero plane above ground, and Z_0 is the surface roughness length parameter. The parameters chosen in the present numerical simulation are $U_\tau = 14.5 \text{ m/s}$, $D = 10 \text{ mm}$, $Z_0 = 1.3 \text{ mm}$. The inlet wind speed corresponding to the building height is $U_H = 14.5 \text{ m/s}$.

As discussed by Tominaga et al. [21], the influence of inflow turbulence for the LES was extremely important. Hence the turbulence intensity profile should be properly reproduced for the accurate simulation of the wind effects on the tall building. The profiles of the mean wind speed and turbulence intensity are shown in Fig.3.

For the case using the inflow turbulence generated by the recycling method, pseudo-periodic boundary conditions are used in the streamwise directions, and the velocity inlet is used for the cases using other inflow generation methods. For all the cases, the periodic boundary conditions are used in the spanwise direction, with all properties being equal at equivalent points on pairs of periodic boundaries which means the distance between the two periodic boundaries

must be such that two-point correlations are zero for all points on a pair of periodic boundaries. This denotes that the distance should be chosen to be at least twice the size of the largest eddies so that the effect of one boundary on the other is minimal. A symmetry condition (vanishing vertical derivatives of the streamwise and spanwise velocity components, as well as vanishing vertical velocity) at the top boundary of the computational domain is adopted. For the LES simulation, outflow boundary conditions are less troublesome, and the familiar zero gradient boundary condition is used for all the variables normal to the outflow boundary. For all the surfaces of the cubic roughness elements, the building model and the ground, the boundary conditions are set to be no-slip wall boundary ($u=v=0$) which is appropriate for the velocity components at solid walls.

4 IMPLEMENTING AND NUMERICAL SETUP

4.1 Subgrid-scale turbulence model

The subgrid-scale (SGS) turbulence models in Fluent code employ the Boussinesq hypothesis as in the RANS models. The simple model was firstly built on Prandtl's mixing length model and it is assumed that we can define a SGS viscosity μ_{sgs} in terms of one length scale and one velocity scale, as suggested by Smagorinsky [22]. However, the Smagorinsky model is purely dissipative which is not particularly appropriate for the high Reynolds number bluff body flows. Germano et al. [23] conceived a procedure in which the Smagorinsky model constant, C_s , is dynamically computed based on the information provided by the resolved scales of motion. The dynamic procedure thus obviates the need for users to specify the model constant C_s in advance. In this study, the dynamic Smagorinsky-Lilly model is adopted as the default subgrid-scale turbulence model, and the C_s obtained by the dynamic Smagorinsky-Lilly model varies in time and space over a fairly wide range. In Fluent code, C_s is clipped between zero and 0.23 by default to avoid numerical instability [16].

4.2 Solution strategy

The solver for incompressible flow in Fluent code employs an algorithm which belongs to a general class of methods called the projection method [24]. In the projection method, the governing equations are solved sequentially. The pressure velocity coupling method chosen for the present simulation is SIMPLC (Semi-Implicit Method for Pressure-Linked Equations Consistent) algorithm [25], which is one of variants of the basic SIMPLE algorithm. For LES, second-order or higher-order discretisation techniques are needed, because upwind differencing, which was standard practice in early CFD computations with RANS turbulence modelling, is far too diffusive and generates large truncation errors. The bounded central difference, a default convection scheme for LES in Fluent code, is used to discretize the convective terms of momentum equations for its relative low diffusivity. The central differencing is also applied to discretize the diffusion term. The time derivative is discretized using the second order backward differences and the spatial discretization is treated implicitly. The non-linear terms produced by fully-implicit discretization of the convection term is solved by an iterative process. The Green-Gauss cell based method is used for numerical approximation of pressure gradients. In order to obtain numerical solutions that are statistically converged quickly, steady RANS model had been used before the LES was performed. In other words, the results of the RANS simulation were adopted to be the initial solutions of the LES.

A 12CPUs workstation was used in parallel for the LES. With consideration of the accuracy of the LES and computational expenses, it was decided to adopt 0.025 seconds per time step for the computations and the convergence criterion for the scaled residuals within one time step is less than 1×10^{-4} .

5 RESULTS AND DISCUSSIONS

5.1 Wind characteristics

Velocity values are taken at the building model position for all the inflow generation methods to validate the characteristics of the generated velocity field when the main domain is computed with the building model filled with additional computing grid points.

In this section, the time-averaged longitudinal velocity and turbulence intensity profiles at the building model position are compared with the target profiles. As demonstrated in Figure.3, the time-averaged velocity and turbulence intensity profiles are reproduced quite well. For the mean velocity profile, all the methods reproduce the variations of velocity in vertical direction quite well, while that generated by DSRFG is slightly higher in the ground neighborhood and lower at the top in comparison with those by the other methods and the target profile. For the turbulence intensity profile, all the three methods generate reasonable results, although there are some discrepancies between the target values and the predictions generated by the RFG and the recycling method which reproduce lower turbulence intensity at higher elevations. Particularly, the turbulence intensity reproduced by the recycling method can impose the random perturbation at the top inlet in the auxiliary driver section for the inflow turbulence generation.

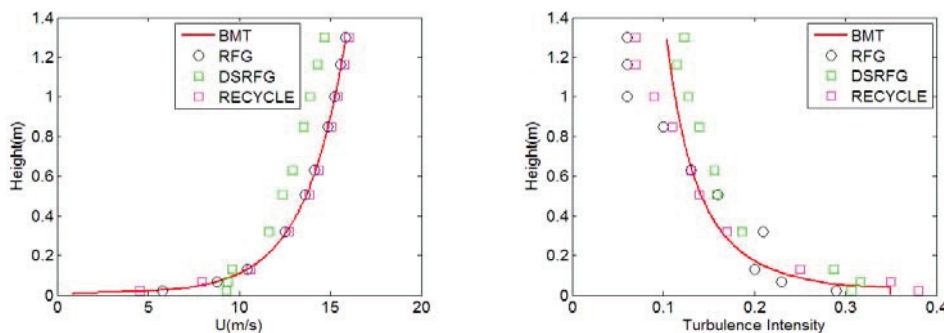


Figure.3 Mean velocity profile and turbulence intensity profile at the building position

The velocity values reproduced by DSRFG at the inlet of the main region are considered herein. In Fig.4, the wind velocity spectral curves at the inlet and the building model position are displayed, which were obtained by two power spectral density estimators: Welch estimator and Yuler-Walker estimator. The spectra of u , v and w velocity fluctuations at the inlet demonstrated in Figs.4(a)~(c) are found to be predicted well in comparison with the target Karman spectrum, while the spectra of three velocity component fluctuations at the building model position (shown in Figs.4(d)~(f)) illustrate relatively large discrepancies in the frequency range larger than 1 Hz. Such a decay of the power spectral density in the relatively higher frequency range was reported to be caused by the grid resolution [4]. Therefore, it is necessary to refine the grids distributed between the building surfaces and the inlet to minimize the decay of the energy contained in the high frequency range.

As discussed in [14] and shown in Figs.5(a)~(c), the velocity component fluctuations generated by RFG are incapable of simulating wind velocities satisfying the von kaman spectrum. Actually, there are obvious discrepancies between the simulated spectra and the target spectra. In particular, the energy contained in the high frequency range is neglected by RFG and such a spectrum is called Gaussian spectrum. For the longitudinal velocity fluctuating component, the spectrum simulated by the recycling method is found to follow the target von Karman spectrum until the frequency reaches to 0.2 Hz, while the energy contained in the higher frequency range shows some discrepancies compared with the von Karman spectrum. Such a decay in the high frequency range may be attributed to the restriction of the grid resolution.

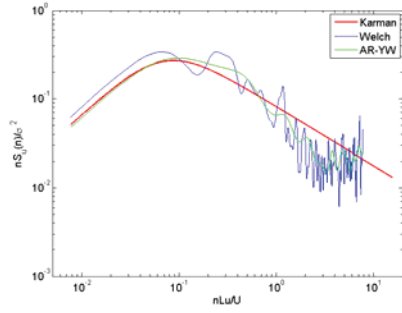


Figure.4 (a) u by DSRFG at inlet

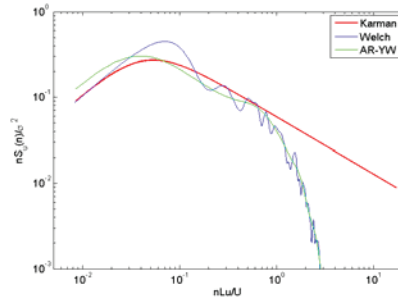


Figure.4 (d) u by DSRFG at building position

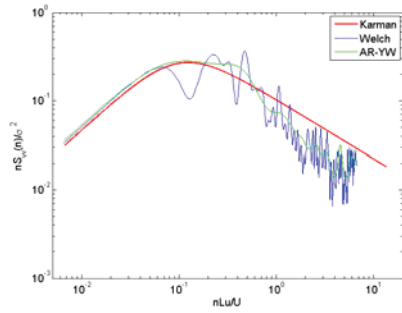


Figure.4 (b) v by DSRFG at inlet

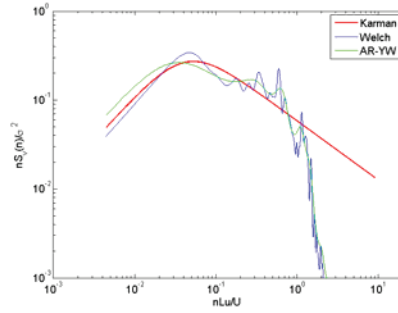


Figure.4 (e) v by DSRFG at building position

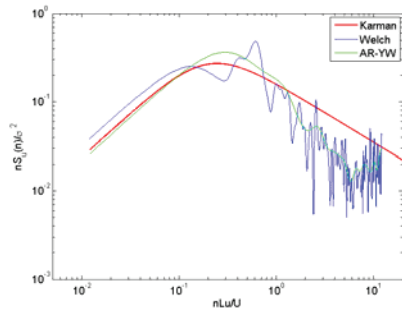


Figure.4(c) w by DSRFG at inlet

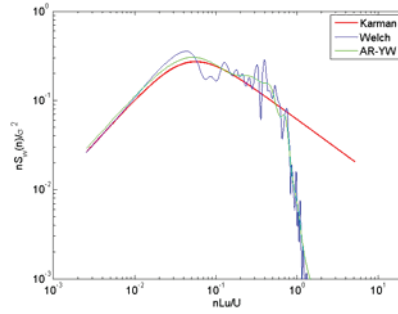


Figure.4 (f) w by DSRFG at building position

Figure.4 Spectra of u, v and w by DSRFG at inlet and building position

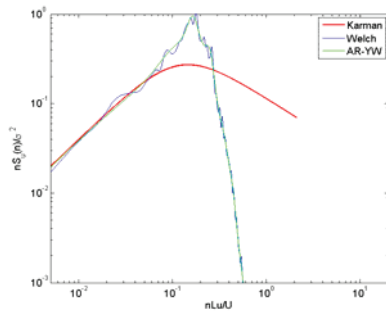


Figure.5(a) u by RFG at building position

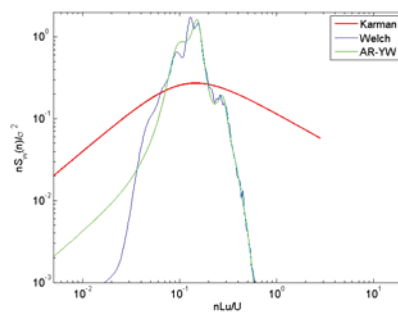


Figure.5(b) v by RFG at building position

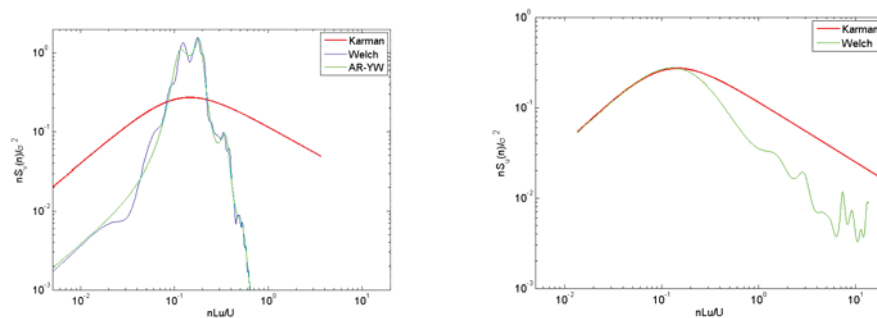


Figure 5(c) w by RFG at building position Figure 5(d) u by the recycling method at inlet
Figure 5 Spectra of u , v and w by RFG at building model position and u by the recycling method at inlet

5.2 Wind loading on the building model

The incident wind flow considered in this study is normal to a side of the building model surface. Fig.6 shows the mean and RMS pressure coefficients at the 2/3 height of the building model, as demonstrated in Fig.6, which were obtained by the present LES and wind tunnel tests. It can be seen that the mean pressure coefficients on the windward face calculated by the three inflow generation methods are nearly the same as those of different wind tunnel experiments. For the side and back faces, the mean wind pressure distribution patterns predicted by different inflow turbulence generation methods are consistent with those of various wind tunnel experiments. For RMS pressure coefficients shown in Fig.6, not only obvious discrepancies exist between the numerical predictions and the wind tunnel test data, but also among the model experimental results. As discussed by Li and Melbourne [26, 27], the reasons for the discrepancies of RMS pressure coefficients among different wind tunnel experiments can be attributed to the different turbulence intensity and turbulence integral length scale values or profiles of approaching wind flows as well as blockage ratio encountered in the model tests.

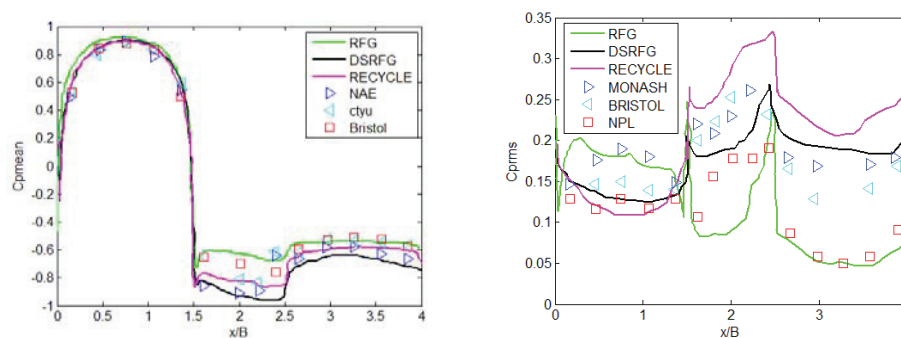


Figure 6 Mean and RMS pressure coefficients at 2/3 height of the building model

As shown in Fig.6, the fluctuating pressure coefficient distribution shapes calculated by the three inflow turbulence generation methods are similar while the distribution predicted by DSRFG is enveloped by the wind tunnel experimental measurements. This illustrates that DSRFG can provide more accurate predictions of the wind loading on the tall building than the other two methods.

6 CONCLUSIONS

In this study, we focused on examining the performance of three methods (DSRFG, RFG and the recycling method) for inflow turbulence generation. The computational results showed that all

the three methods can reproduce the time-averaged velocity and turbulence intensity profiles at the building position quite well. The spectra of random wind velocities reproduced by DSRFG showed good agreement with the target spectrum at the inlet and that of the longitudinal velocity simulated by the recycling method is similar to the von-Karman spectrum. However, RFG is incapable of generating fluctuating velocity satisfying the prescribed von Karman spectrum. The velocity spectra at the building position simulated by DSRFG and the recycling method are affected by the grid resolution between the windward surface of the building and the inlet. The lower grid resolution may result in fast decay of wind velocity spectra in the high frequency range. In order to verify the accuracy of the wind loading predication by the three methods, the time-averaged and RMS pressure coefficients distributed at the 2/3 height of the CAARC building were compared with available wind tunnel test data obtained by several labs. It was found that the mean pressure coefficients on the front surface of the building were accurately predicted by all the three inflow generation methods and the pressure coefficient distribution pattern on the side and back surfaces were also well predicated. All the RMS pressure coefficients obtained by the three inflow turbulence generation methods fall in the range of the model test data by various wind tunnel experiments, verifying that the effects of turbulence characteristics of approaching wind flows on the building surface pressures were reproduced to some extent. It was shown through the comparison that DSRFG can provide more accurate predictions of the wind loading on the tall building than the other two methods.

7 ACKNOWLEDGEMENT

The work described in this paper was fully supported by a grant from the Research Grants Council of Hong Kong Special Administrative Region, China (Project No: CityU 117709) and a grant from City University of Hong Kong (Project No. 7002615). The authors would like to thank Dr. Shenghong Huang of the University of Science and Technology of China for helpful discussions.

8 REFERENCE

- 1 Tamura T. Towards practical use of LES in wind engineering. *J Wind Eng Ind Aerodyn* 2008;96:1451-71.
- 2 Spalart PR, Leonard A. Direct numerical simulation of equilibrium turbulent boundary layers. *5th Symposium on Turbulent Shear Flows* 1985;1:9.
- 3 Lund S, Wu X, Squires KD. Generation of Turbulent Inflow Data for Spatially-Developing Boundary Layer Simulations. *Journal of Computational Physics* 1998;140:233-58.
- 4 Kataoka H, Mizuno M. Numerical flow computation around aeroelastic 3 D square cylinder using inflow turbulence. *WIND STRUCT INT J* 2002;5:379-92.
- 5 Nozawa K, Tamura T. LES of the flow around a low-rise building immersed in a rough-wall turbulent boundary layer. *J Wind Eng Ind Aerodyn* 2002;90:1151-62.
- 6 Liu K, Pletcher RH. Inflow conditions for the LES of turbulent boundary layers: a dynamic recycling procedure. *Journal of Computational Physics* 2006;219:1-6.
- 7 Hoshiya M. Simulation of multi-correlated random processes and application to structural vibration problems. *Proceedings of JSCE* 1972;204:121-8.
- 8 Iwatani Y. Simulation of multi-dimensional wind fluctuations having any arbitrary power spectra and cross spectra. *Journal of Wind Engineering* 1982;11:5-18.
- 9 Kondo K, Murakami S, Mochida A. Generation of velocity fluctuations for inflow boundary condition of LES. *J Wind Eng Ind Aerodyn* 1997;67:51-64.
- 10 Perret L, Delville J, Manceau R, Bonnet J P. Generation of turbulent inflow conditions for LES from stereoscopic PIV measurements. *Int J Heat Fluid Flow* 2006;27:576-84.

- 11 Smirnov A, Shi S, Celik I. Random flow generation technique for LESs and particle-dynamics modeling. *Journal of Fluids Engineering* 2001;123:359.
- 12 Klein M, Sadiki A, Janicka J. A digital filter based generation of inflow data for spatially developing direct numerical or LESs. *Journal of Computational Physics* 2003;186:652-65.
- 13 Batten P, Goldberg U, Chakravarthy S. Interfacing statistical turbulence closures with large-eddy simulation: Boundary conditions for LES. *AIAA J* 2004;42:485-92.
- 14 Huang S H, Li Q S, Wu J R. A general inflow turbulence generator for LES. *J Wind Eng Ind Aerodyn* 2010;98:600-17.
- 15 Kraichnan R H. Diffusion by a random velocity field. *Phys Fluids* 1970;13:22.
- 16 Fluent 6.3 user's guide. Fluent Inc 2006.
- 17 Kataoka H. Numerical simulations of a wind-induced vibrating square cylinder within turbulent boundary layer. *J Wind Eng Ind Aerodyn* 2008;96:1985-97.
- 18 Melbourne W.H. Comparison of measurements on the CAARC standard tall building model in simulated model wind flows. *J Wind Eng Ind Aerodyn* 1980;6:73-88.
- 19 Obasaju E. Measurement of forces and base overturning moments on the CAARC tall building model in a simulated atmospheric boundary layer. *J Wind Eng Ind Aerodyn* 1992;40:103-26.
- 20 Goliger A, Milford R. Sensitivity of the CAARC standard building model to geometric scale and turbulence. *J Wind Eng Ind Aerodyn* 1988;31:105-23.
- 21 Tominaga Y, Mochida A, Murakami S, Sawaki S. Comparison of various revised k- ϵ models and LES applied to flow around a high-rise building model with 1: 1: 2 shape placed within the surface boundary layer. *J Wind Eng Ind Aerodyn* 2008;96:389-411.
- 22 Smagorinsky J. General circulation experiments with the primitive equations. *Mon Weather Rev* 1963;91:99-164.
- 23 Germano M, Piomelli U, Moin P, Cabot WH. A dynamic subgrid-scale eddy viscosity model. *Physics of Fluids A: Fluid Dynamics* 1991;3:1760.
- 24 Chorin A J. Numerical solution of the Navier-Stokes equations. *Math.Comp* 1968;22:745-62.
- 25 Van Doormaal J, Raithby G. Enhancements of the SIMPLE method for predicting incompressible fluid flows. *Numerical Heat Transfer* 1984;7:147-63.
- 26 Li Q S, Melbourne W.H. The effect of large-scale turbulence on pressure fluctuations in separated and reattaching flows. *J Wind Eng Ind Aerodyn* 1999;83:159-69.
- 27 Li Q S, Melbourne W.H. An experimental investigation of the effects of free-stream turbulence on streamwise surface pressures in separated and reattaching flows. *J Wind Eng Ind Aerodyn* 1995;54:313-23.