Research Field: Outdoor environment

Research Year: FY2021
Research Number: 20212013
Research Theme: Study on effects of morphological parameters on urban ventilation by wind tunnel experiments and LES calculations

Representative Researcher: LI Biao

Budget [2021 FY]: 475000Yen

*There is no limitation of the number of pages of this report.
*Figures can be included to the report and they can also be colored.
*Submitted reports will be uploaded to the JURC Homepage.

1. Research Aim

This research from the perspective of urban morphology, uses the combination of wind tunnel experiment and numerical simulation to deeply explore the influence of the non-uniformity of urban buildings on the aerodynamic characteristics and mechanism of pollutant dispersion in the boundary layer. It provides a combined experiment and LES calculation study for the evaluation of the drag effect and pollutant dispersion effect of the non-uniform buildings by morphological modelling methods.

- Wind tunnel experiment explores the influence of complex buildings

Based on the large-scale boundary layer wind tunnel experiment, the influence of four shape parameters on the flow characteristics and the overall drag force of the buildings is deepened. The distribution of local form drags and their contribution to the overall drag are systematically investigated, and the influence mechanism of several shape parameters on the drag coefficient of buildings is explored.

- LES calculation on the flow mechanism and morphological effects of complex buildings

Based on the high-precision LES numerical simulation method, the influence of complex buildings on the atmospheric flow is deeply studied. Combined with the wind tunnel experiment, a relatively complete mechanism of horizontal spatial heterogeneity will be formed.

The outcome of this study will provide experimental and theoretical support for aerodynamic parameters estimation on non-uniform buildings in the real urban area, increase the accuracy of the urban canopy models, and improve the precision of the urban weather forecast and the pollutant diffusion calculation.

2. Research Method

2.1 Computational Domain and Grid

This study establishes a numerical wind tunnel simulation model for buildings identical to the wind tunnel tests, as shown in Figure 3, it consists of test platforms and wind tunnel bodies around inhomogeneous buildings, ramp transition sections. The size of the calculation domain is below the limits of computer computing power, staccble turbulent inflow, sufficiently long test sections to ensure fully developed outflows and true wind tunnel test details.

For the calculation of the RANS method, the final size of the calculation domain is X×Y×Z=7000×4000×3000mm, and the upstream domain length is 19H=1995mm, where H represents the height of the building model. Creating a computing grid using hexahedron units allows better computational domains for structured grids, resulting in a fine hexahedron grid, which is verified by grid independence and ultimately has a hexahedron grid of approximately 2 million. The heights of adjacent units on the first wall of all four walls of the test platform wind tunnel are descended.Fine adjustment to determine y+ requirements of different codes. In addition, y+ shall meet the requirements of different wall functions. [6][11]

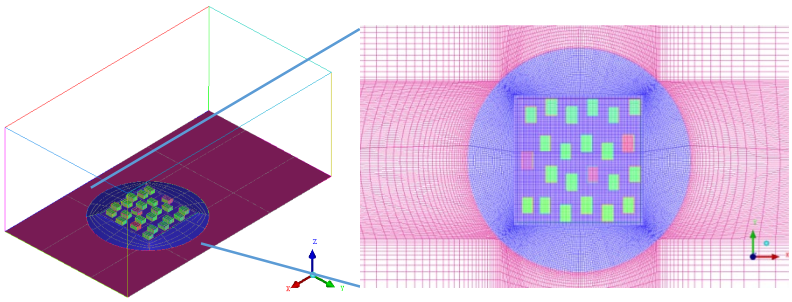

Fig. 1. Computational domain and grid details near buildings.

For the calculation of LES method, the internal mapping method (Tabor & Baba-Ahmadi, 2010) is used in this study to generate the turbulent inflow data of the neutral boundary layer for LES calculation. Therefore, the solution domain consists of two regions, as shown in the figure below: 1) the driving area, in which the flow rate is extracted from the cross section at a specific downstream location and recovered to the inlet to generate the velocity fluctuation in the model area;2) The model area used to simulate the flow on the target model. The precursor simulation in Figure 4 with horizontal circulation boundary and the same terrain as the driving area initializes the solution area. The wind profile obtained from the wind tunnel experiments initializes the precursor simulation.

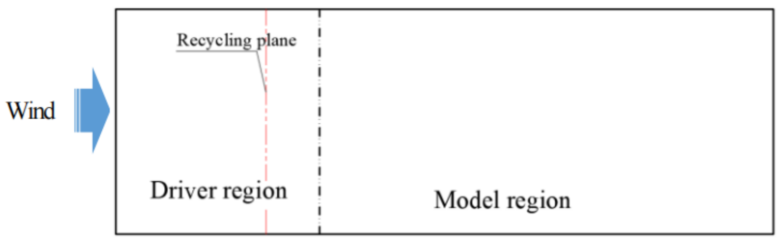

Fig. 4 Solution domain setup

The dimensions of the calculation domain are 9.6 m (91.4 H), 3.84 m (36.6 H) wide and 1.59 m (15.2 H) high.The upwind fetch from the last row of the building models in the driver region to the first obstacle row of the target building array is.24H. The distance from the final row of target buildings to the exit boundary is 29H and the outer boundary is 9H. The above domain sizes meet the main simulation requirements recommended by Tominaga et al. (2008). The drive zone roughness elements are 120mm (long) 120mm (wide) 100mm (high), staggered and have a plane area density of 16%.

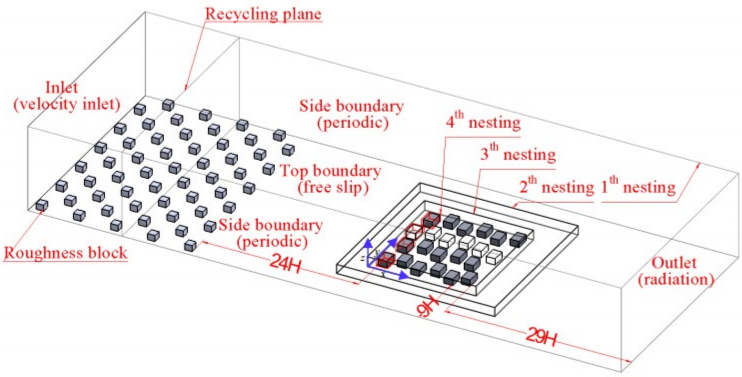

Fig. Nested grid configuration

2.2 Large eddy simulation

The recycling method is used to generate inflow turbulent data for LES in a neutral boundary layer. Internal mapping, being one implementation of the recycling method, is adapted to this paper. The solution domain of the simulation is composed of two regions: (1) a driver region, where flow quantities are extracted from a cross section at a certain downstream location and recycled into the inlet, to generate the velocity fluctuations for the model region (2) a model region for simulating flow over the target model.

The simulation is initialized with turbulent 3-dimensional velocities which are mapped from a precursor simulation with cyclic boundaries and same topography as the driver region of the solution domain. Besides, the constant volume is conserved and the wind profile from the wind tunnel experiments initializes the precursor simulation.

The instantaneous wind velocities of the simulation at the inflow of the driver region are calculated as follows,

$$u_{inlet} = U_z + \varphi \times [u - U_{av}]_{rec} \quad (1)$$

$$v_{inlet} = \varphi \times [v - V_{av}]_{rec} \quad (2)$$

$$w_{inlet} = \varphi \times [w - W_{av}]_{rec} \quad (3)$$

$$\varphi = 0.5\left\{1 - \frac{\tanh\left[\frac{8.0(\theta - 1.0)}{4.0(0.3 - \theta) + 0.7}\right]}{\tanh(8.0)}\right\} \quad (4)$$

where U_z is stream-wise velocity which is averaged horizontally and temporally from the simulation results of the precursor run. U_{av}, V_{av}, W_{av} are temporally averaged stream-wise velocities. u, v, w are instantaneous velocity components. The subscript 'inlet' and 'rec' denote the variables at the inlet and the recycling planes. $\theta = z/\delta$, δ is the boundary layer thickness. φ is the damping factor.

The *PALM* (**PA**rallelized **L**ES **M**odel), open source code developed by Leibniz University Hannover is used in this paper. It is especially designed for performing on massively parallel computer architectures, and so far have tested up to 32,000 cores. *PALM* has been widely used in many research fields, such as the turbulent airflow in street canyon, the turbulent structure of urban boundary layer, urban wind environment and pollutant dispersion. *PALM* adopts LES turbulence model, in which eddies smaller than cut-off length is modelled while large eddies are directly resolved. The filtered Navier-Stokes equations are as follows:

$$\frac{\partial \overline{u_i}}{\partial x_i} = 0 \tag{5}$$

$$\frac{\partial \overline{u_i}}{\partial t} + \overline{u_j}\frac{\partial \overline{u_i}}{\partial x_j} = -\frac{1}{\rho}\frac{\partial \bar{p}}{\partial x_j} - \frac{\partial}{\partial x_j}\left(\upsilon \frac{\partial \overline{u_i}}{\partial x_j}\right) - \frac{\partial \tau_{ij}}{\partial x_j} \tag{6}$$

where u_i and u_j are the velocity components, ρ and υ are density and viscosity. The overbar symbol denotes the filtering operator, and τ_{ij} is the subgrid scale stress (SGS) term defined as:

$$\tau_{ij} = \overline{u_i u_j} - \overline{u_i}\,\overline{u_j} \tag{7}$$

The Boussinesq hypothesis is used to calculate the SGS stress term, and a modified version of Deardorff's subgrid-scale model is adopted as follows,

$$\tau_{ij} - \frac{2}{3}e\delta_{ij} = -\upsilon_{SGS}\left(\frac{\partial \overline{u_i}}{\partial x_j} + \frac{\partial \overline{u_j}}{\partial x_i}\right) \tag{8}$$

$$\upsilon_{SGS} = C_m l e^{\frac{1}{2}} \tag{9}$$

where υ_{SGS} is the SGS turbulent viscosity; e is the SGS turbulent kinetic energy (TKE); C_m of 0.1 is the model constant, l is the SGS mixing length. For neutral boundary layer, it depends on the altitude from the wall z, grid spacing, and is calculated as follows:

$$l = \min(1.8z, (\Delta x \Delta y \Delta z)^{\frac{1}{3}}) \tag{10}$$

Moreover, the model closure includes a prognostic equation for the SGS-TKE e, under neutral boundary layer, is calculated as follows,

$$\frac{\partial e}{\partial t} = -\overline{u_j}\frac{\partial e}{\partial x_j} - \tau_{ij}\left(\frac{\partial u_i}{\partial x_j}\right) + \frac{\partial}{\partial x_j}\left(2\upsilon_{SGS}\frac{\partial e}{\partial x_i}\right) - \epsilon) \tag{11}$$

where ϵ is the dissipation rate within a grid volume,

$$\epsilon = C_\epsilon \frac{e^{\frac{1}{2}}}{l} \tag{12}$$

The coefficient C_ϵ is specified as 0.93. To reduce the impact of the outflow on the inner flow state, all the variables normal to the outflow boundary are set as zero gradient. Periodic boundary condition is used for the span-wise of the computational domain, and free slip condition is used as the top boundary condition. Wall functions are based on Monin-Obukhov similarity theory, so the sub-grid stress of the first vertical layer τ_{i3} is calculated as below：

$$\tau_{i3} = -\left[\frac{\overline{u_r}}{log(z_s/z_0)}\right]^2 \frac{\overline{u_\iota}}{\overline{u_r}} \quad (13)$$

where, z_0 is roughness length, κ is Von Karman constant, being set as 0.4; z_s is the altitude at the first vertical grid; z_0 is set 0.0005 m (0.1 m in full scale); $u_r = (u^2 + v^2)^{1/2}$ is the resolved resultant horizontal velocity at the first vertical grid.

2.3 **RANS Model**

standard k－ε model

The k-**ε** model is the simplest turbulence model, providing only initial or boundary conditions. It also provides excellent performance for many industrial processes. The governing equation is:

$$\frac{\partial(ku_i)}{\partial x_i} = \frac{1}{\rho}\frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right] + \frac{P_k}{\rho} - \varepsilon$$

$$\frac{\partial(\varepsilon u_i)}{\partial x_i} = \frac{1}{\rho}\frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_\varepsilon}\right)\frac{\partial \varepsilon}{\partial x_j}\right] + C_{\varepsilon 1}\frac{\varepsilon}{k}\frac{P_k}{\rho} - C_{\varepsilon 2}\frac{\varepsilon^2}{k}$$

Where P_k is the production of turbulent flow energy, μ_t is eddy viscosity. They are given by equations 1 and 2 respectively.

$$P_k = -\rho\overline{u_i'u_j'}\frac{\partial u_j}{\partial x_i} = \mu_t \cdot 2S_{ij}S_{ij}$$

$$\mu_t = \rho C_\mu \frac{k^2}{\varepsilon}$$

Where S_{ij} is the strain rate tensor and σ_k, $C_{\varepsilon1}$,$C_{\varepsilon2}$,C_μ is the model constant, the values are usually 1.0, 1.11, 1.92 and 0.09 respectively. Note that $\sigma_\varepsilon$varies from the default value of 1.44-1.11 to simulate the neutral atmospheric boundary layer according to studies provided by Hargreaves and Wright.

Standard k - ω model

Standard k - ω Model was originally a two equation model proposed by Wilcox to consider low Reynolds number, compressible and shear flow propagation. The model is a low Reynolds number model, which has high sensitivity to reverse pressure gradient and can adapt to rough initial turbulent flow field.

The turbulent viscosity coefficient equation is:

$$\mu_t = \rho C_\mu k/\omega$$

where, $\omega = \varepsilon/k$。

The turbulent fluctuating kinetic energy k equation is:

$$\frac{\partial(\rho k)}{\partial t} + \frac{\partial(pku_j)}{\partial x_j} = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_i}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right] + \rho P - \rho\omega k$$

The special dissipation ω equation of turbulence is:

$$\frac{\partial(\rho w)}{\partial t}+\frac{\partial(pwu_j)}{\partial x_j}=\frac{\partial}{\partial x_j}\left[\left(\mu+\frac{\mu_t}{\sigma_w}\right)\frac{\partial w}{\partial x_j}\right]+C_{1w}\frac{\rho Pw}{k}-C_{2w\rho}\rho\omega^2$$

Where, ρ is the density, ε is the turbulent dissipation rate, and t is the time; x_j is the coordinate vector; u_j is the velocity component;P is the pressure term, $C_\mu, C_{1\omega}, C_{2\omega}$ is the empirical constant, where,$C_\mu = 0.09, C_{1\omega} = 0.555, C_{2w} = 0.833$;$\sigma_k$ and $\sigma_\omega$are the Prandtl numbers corresponding to the turbulent fluctuating kinetic energy$k$and the turbulent special dissipation rate ω, where $\sigma_k = 2\ 0, \sigma_w = 2.0$。

Additionally, SA , RNG k- ε , Realizable k-ε, SST k- ω turbulence models are also studied in our work, in order to save space, the relevant equations are no longer described.

2.4 Wind tunnel experiments

The atmospheric boundary layer wind tunnel experiments were performed in the small section of the Wind Tunnel and Wave Trough Laboratory of Harbin Institute of Technology. The experimental design diagram is shown in Fig. 1. Following the load code for the design of building structures of China (*GB 50009-2012*), the turbulent inflow arrangement adopts the standard *C*-type terrain, which represents the approaching flow over the dense areas such as trees, gentle hilly land, low-rise buildings and high-rise buildings. The terrain roughness index in power-law profile is 0.22. The fetch length is 19.2 m, and the thickness of the boundary layer is about 1.3 m. The tested building array is shown schematically in Fig. 2, with geometric scale of 1/200. The building array consists of total 23 cuboids with uniform size of 213.7mm (length) × 144.4mm (width) × 105mm (height). The plane area density is 27.5%, and frontal area density is 25%. The blockage ratio is 1.9%. For concise, the $B_{i,j}$ is used to represent building model, where *i* represents the row number coordinate, and *j* represents the column number coordinate. The pressure taps were arranged on the six white blocks $B_{i,3}$ along the wind direction in Fig. 2.

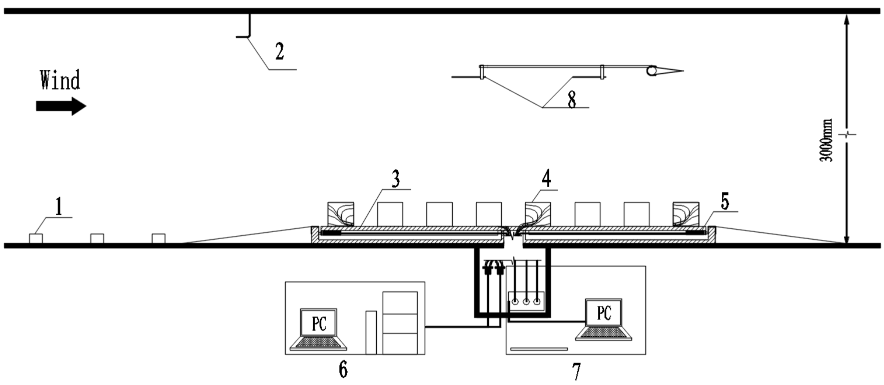

Fig. 1 Schematic diagram of wind tunnel platform. 1Terrain roughness elements, 2 Pitot-static tube, 3 Floating device, 4 Pressure module, 5 Force sensor, 6 Pressure scanning valve system, 7 Force sensor system. 8 Hot-wire anemometers.

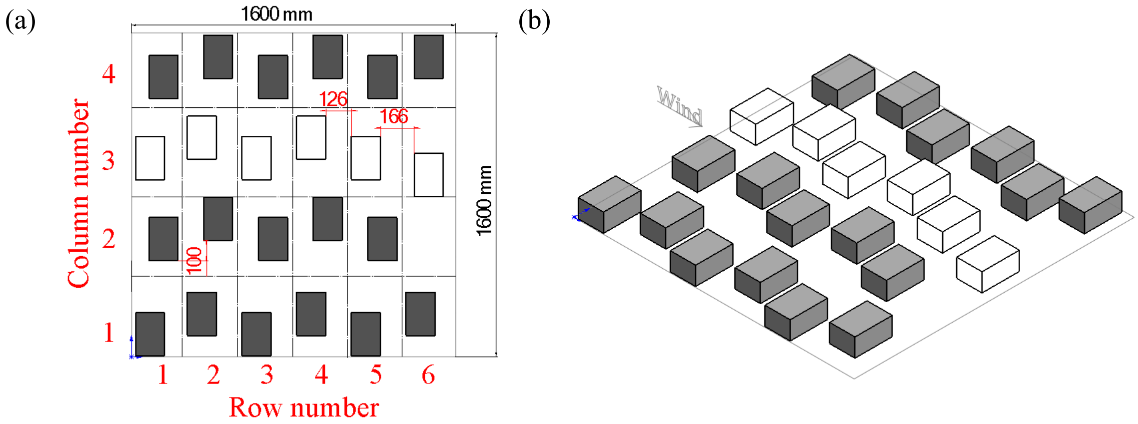

Fig. 2 Schematic diagrams of block arrangements. (a) Plan view (b) Isometric view. The white blocks are used for pressure measurement. The array comprises 6 rows × 4 columns, with the absence of one building model at the sixth row and second column, total 23 blocks.

The atmospheric boundary layer wind tunnel experiment was carried out in a low-speed test section of the wind tunnel of Wind Engineering Research Center of Shijiazhuang Tiedao University. The rectangular section of the experimental section is 4.4 m wide and 3 m high, and the length of small experimental section is 24 m. The maximum wind velocities that the wind tunnel are capable of producing are greater than 30.0 m/s.

3. Research Result

It is proposed to use the research method of combining wind tunnel experiment and numerical simulation to build the research of the non-uniform buildings' influence on the urban ventilation and pollutant dispersion. The technical route is as follows: 1) Based on the existing wind tunnel experimental design and data (Fig. 3), a similar model experiment scheme is setup for TPU wind tunnel; 2) Carry out the design and construction of model with complete geometric similarity; 3) Measure the flow field and pollutant distribution in wind tunnel under different non-uniformity configurations; 4) The detailed LES simulation is carried out (Fig. 4), and the dynamic mechanism analysis of urban ventilation is discussed by comparing the wind tunnel experiment results.

The turbulent characteristics are examined without and with the immersed buildings. The average values of the four points of the 400 mm×400 mm square in the center of the model area are taken as the spatial average. Flow characteristics obtained from LES and experiment without the immersing building array are shown in Fig. 5. It can be concluded that the mean wind velocity as well as the turbulence intensity of stream-wise velocity component I_u are consistent with the experimental results.

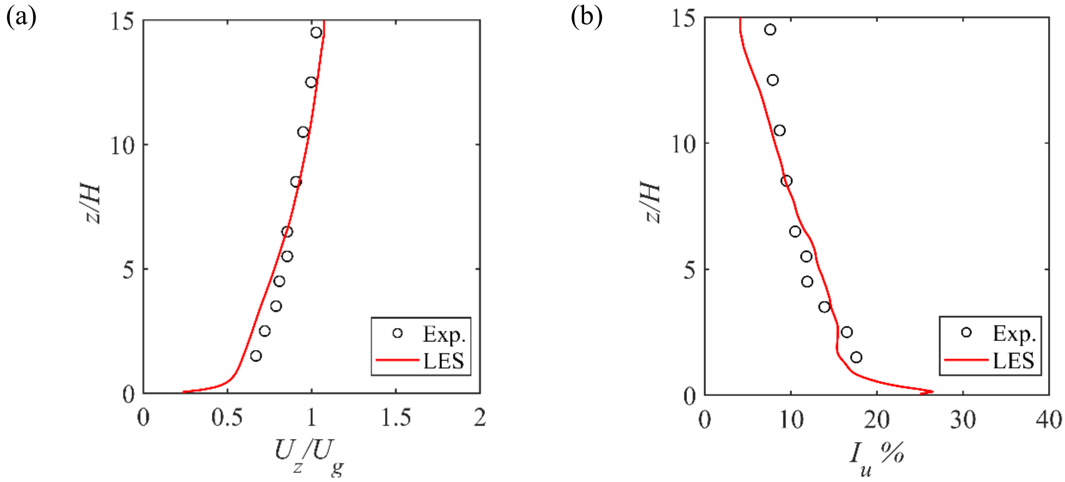

Fig. 5. Approaching flow characteristics obtained from LES and experiment without immersing building models. (a) Normalized stream-wise mean velocity U_z/U_g, where U_g is the mean wind velocity at boundary layer height δ. (b) Stream-wise turbulence intensity I_u.

To investigate the flow characteristics for the flow over horizontal non-uniform buildings, *Q*-criterion is adapted, shown in Fig. 6. We can see that the instantaneous vortex structures present distinct randomness and fragmentation. For the first row of buildings, the horseshoe-type vortex at the windward face can be clearly observed. In addition, the incoming flow clearly separates at the roof of the first row of buildings, therefore creating strong shear stress. The columnar vortexes formed by the shear layer are transported downstream, resulting in a higher velocity near the roof than in other areas.

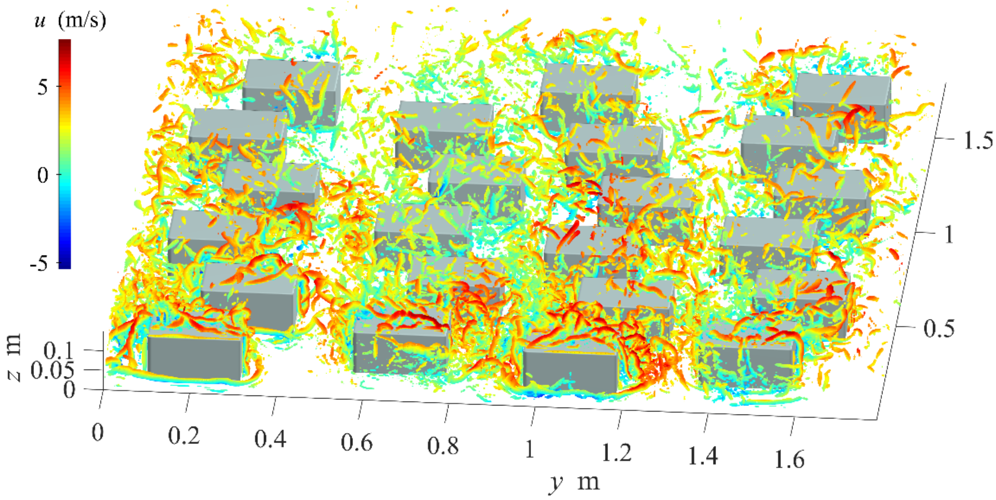

Fig 6. Instantaneous iso-surface of *Q* criterion, where *Q* equals to 3×10^4. The color chart shows magnitude of instantaneous velocity.

The objective of this proposal is to explore the urban ventilation and pollutant dispersion of different configurations of non-uniform buildings, using the comparison study among LES calculation, pollutant dispersion measurement in wind tunnel in TPU, and a previous wind tunnel measurement. The outcome of this study will provide experimental and theoretical support for aerodynamic parameters estimation on non-uniform buildings in the real urban area, increase the accuracy of the urban canopy models, and improve the precisions of the urban weather forecast and the pollutant diffusion calculation.

4. Published Paper etc.

[Underline the representative researcher and collaborate researchers]

[Published papers]

1. Wang, L.; Liu, J.; Jiang, C.; Li, B.; Song, D.; Lu, M.; Xuan, Y. Large-Eddy Simulations on the Effects of Two Wind Passage Types between Buildings on the Airflow and Drag Characteristics. Atmosphere 2021, 12, 1646. https://doi.org/10.3390/atmos12121646

2. Jiang C., Li B., Zhou D, Wang L., Xuan Y., Liu J., Evaluation of CFD Simulation Using RANS and LES Models on Wind Tunnel Experiments, JWEIA, manuscript.

[Presentations at academic societies]

3. Studies on roughness evaluation with non-uniform buildings area, Research Seminar (Online)，2022.2.22, JURC, Japan
4.

[Published books]

1.
2.

[Other]

Intellectual property rights, Homepage etc.

5. Research Group

1. Representative Researcher

LI Biao

2. Collaborate Researchers

1. Zhou Deli
2. JIANG Cunyan
3. LIU Jing
4. XUAN Yingli

6. Abstract (half page)

Research Theme:

Study on effects of morphological parameters on urban ventilation by wind tunnel experiments and LES calculations

Representative Researcher (Affiliation)

LI Biao*, ZHOU Deli, LIU Jing, JIANG Cunyan (Harbin Institute of Technology)

XUAN Yinli (Tokyo Polytechnic Unversity)

This research from the perspective of urban morphology, uses the combination of wind tunnel experiment and numerical simulation to deeply explore the influence of the non-uniformity of urban buildings on the aerodynamic characteristics and mechanism of pollutant dispersion in the boundary layer. It provides a combined experiment and LES calculation study for the evaluation of the drag effect and pollutant dispersion effect of the non-uniform buildings by morphological modelling methods. The outcome of this study will provide experimental and theoretical support for aerodynamic parameters estimation on non-uniform buildings in the real urban area, increase the accuracy of

the urban canopy models, and improve the precision of the urban weather forecast and the pollutant diffusion calculation.