Buoyancy flows and pollutant dispersion through different scale urban areas: CFD simulations and wind-tunnel measurements

Peng-Yi Cui^a, Zhuo Li^a, Wen-Quan Tao^{a,b,*}

Abstract

In this paper, a coupled CFD model was established to study the multiscale problems on the mixed force and buoyancy flow and dispersion passing neighborhood scale – street scale – indoor scale models, and the numerical results were validated by wind-tunnel measurements with Richardson numbers (Ri) from 0 to 4.77 with SF_6 being the tracer gas. The basic flow, heat and pollutant transfer were solved with the 3-D steady RANS (Reynolds-Averaged Navier-Stokes) equations. The results show that when $Ri \leq 0.85$, the standard $k-\varepsilon$ model (SKE) can better predict the flow and temperature fields. When $Ri > 0.85$, the realizable $k-\varepsilon$ model (RLKE) perform better. For the same turbulence model equation, with the increasing Ri the effects of the two near-wall functions (standard and non-equilibrium wall functions, SWFs and NEWFs) on the flow structures and temperature distributions become more and more significant. The specific value of Sc_t has a significant effect on predicting the pollutant dispersion and the optimal value is 0.7 for the studied cases. It is also found that for indoor flow caused by an outdoor street flow there also exists the Re -independence region. For the model studied only when $Re_H \geq 7.57E+03$, the wind-tunnel measured results can represent the realistic cases

meaningfully for both flow and pollutant distributions in street canyon and inside the room.

Keywords: Wind tunnel; Computational fluid dynamics (CFD); Buoyancy flow; Pollutant dispersion; Multiscale problem

1. Introduction

The physical phenomena that mixed forced and buoyancy flow with harmful pollutants passes over different geometry scale urban areas and enters into indoor environment to affect the health of residents belong to multiscale physical problems [1,2]. Recently, the frequent haze weather caused by the increasing level of air pollutants such as PM_{10} and $PM_{2.5}$ has caught many people's attentions in many big cities of China, which is mainly resulted from the coal combustion, industrial and traffic exhausts [3-6]. Besides, the city managers and planners also pay attentions to the diffusion of air pollutants due to incidents such as terrorist attack and accidental leakage of toxicity fluids in different scale urban areas [7,8]. The studied regions can be divided into several ranges of length scales: regional scale (up to 100 or 200 km), city scale (up to 10 or 20 km), neighborhood scale (up to 1 or 2 km) and street scale (less than 100 ~ 200 m) [9], indoor scale (less than 10 ~ 20 m) and even body scale (less than 1 ~ 2 m). However, most of the previous studies mainly concentrated on the same geometry scales, and three approaches have been used: (1) on-site full-scale experiments; (2) reduced-scale physical modeling (wind-tunnel or water-channel experiments); and (3) numerical simulations based on CFD techniques.

There are pros and cons to these three research methods for studying flow and dispersion within urban environment. CFD method can provide detailed information under well-controlled conditions, but the model and method should be validated by some test results. Reduced-scale model can give some accurate experimental data, but is required to overcome the similarity constraints, especially when the forced flow and buoyancy effects are combined together. More intuitive and real data can be obtained from on-site experiments. However, for this method, apart from huge sources required, the experimental repeatability is quite small due to the uncontrollability of the meteorological conditions. van Hoof and Blocken [10] carried out the full-scale measurements of indoor environmental conditions and natural ventilation in a large semi-enclosed stadium and discussed four issues: (1) the advantage and disadvantage of full-scale on-site measurements versus reduced-scale wind tunnel measurements; (2) the limited number of measurement positions inside the stadium; (3) the repeatability of the full-scale measurements; and (4) the possibilities and limitations of the full-scale measurements for CFD validation. Given the above factors, most researchers usually conducted their studies by coupling CFD methods with other two methods for the CFD validation. Gousseau et al. [11] evaluated the performance of two different modeling methods (RANS standard $k-\varepsilon$ model and LES) applied to predict near-field pollutant dispersion in an actual urban environment: downtown Montreal, validated by the detailed wind-tunnel experiments. They found that LES with the dynamic subgrid-scale model shows a better performance without requiring any parameter input to solve the dispersion equation. Allegrini et al. [12] used the wind-tunnel measurements to validate the scaled CFD model for the mixed buoyancy flow in urban street canyons, and the numerical results showed that a significantly better agreement can be achieved by using a standard

71 k - ε model with the NEWFs (non-equilibrium wall functions) than LRNM (low-Reynolds
72 number modeling). Stavrakakis et al. [13] applied CFD method coupled with the on-site
73 building-scale experiments to investigate the natural cross-ventilation and evaluate the
74 thermal comfort inside the building. They concluded that the RNG (renormalization
75 group) k - ε model performed relatively better, especially for temperature predictions and
76 hence they chosen it for further thermal comfort estimation purposes.

77 Obviously, the above studies mainly focused on one geometric scale (neighborhood
78 scale, street scale or building scale). However, the flow and dispersion within urban
79 environment belong to multiscale phenomena, the geometry scales of which are always
80 across more than three orders of magnitude, taking the pollutant transport in Lujiazui
81 region of Pudong district, Shanghai (see Fig. 1) as an example. The pollutant transport
82 process in such an area could be described as: the pollutants are released into the ambient
83 air in somewhere of the upwind direction and diffuse with the air flows that are affected
84 by many factors, then pass through building groups in the neighborhood scale, and arrive
85 at a high-rise building located in a street canyon, finally enter into a room through the
86 open windows, and affect the health of residents. In general, there are three numerical
87 methods to solve the above multiscale problem to obtain the detailed information in the
88 smallest scale geometry. The first one is called “coupled” CFD simulation or full-scale
89 simulation, in which grid generation and equation solving are conducted simultaneously
90 within the whole computational domain. This method allows the proper calculation of air
91 flow in the smallest scale geometry, but needs a high-resolution grid system and a
92 relatively high computational cost for a large grid number [14]. The second method is
93 called “de-coupled” simulation, in which the computational domain is divided into

several regions according to the different geometry scales, then the separate simulations are conducted, for example, one for the outdoor flow and other for the indoor flow, each in their own computational domain [15-17]. In the outdoor simulation, the information (generally pressure coefficients) is obtained at the closed ventilation openings to be used as the boundary condition for the indoor simulation. The accuracy of this method can easily be compromised due to the simplifications involved. To improve the second approach, another “de-coupled” method called “from top-to-down solution procedure” is proposed with increasing grid fineness [18,19]. Both the two “de-coupled” methods require an accurate full-scale simulation to validate and evaluate their simulation results.

It is well-known that the wind-tunnel experiment is an important tool to investigate the basic flow, heat and mass transfer within urban environment and can also provide data for the validation of the CFD method. However, to the authors’ knowledge, the above complex multiscale problems have been only investigated by on-site full-scale measurements coupled with CFD simulations by van Hooff and Blocken [14] and not yet performed by reduced-scale wind tunnel measurements. Their study presented a coupled CFD modeling method for urban wind flow and indoor natural ventilation on a high-resolution body-fitted grid using realizable $k-\varepsilon$ model and the CFD model was validated by the on-site full-scale measurements [14]. Then, they applied this coupled CFD modeling to investigate the effect of direction and urban surroundings on the ACH of a large semi-enclosed stadium [20]. For the reduced-scale wind-tunnel measurements in order to obtain meaningful results of the multiscale problems mentioned above there are two important challenges. The first challenge is the satisfaction of flow similarity between prototype and wind tunnel. In this regard previous studies [21, 22] have shown

that for flows over buildings once Reynolds number of the oncoming flow is larger than a certain values, denoted by Re_{crit} , the values of Re will not affect the flow structure too much. The value of Re_{crit} is usually 2-3 orders of magnitude less than Re of the prototype flow. When Re is larger than Re_{crit} flow is in the Re -independence region. The existence of Re -independence region creates a very favorable condition for adopting small-scale wind tunnel test. Second, it should be guaranteed that the smallest scale geometry of the multiscale modeling satisfies the Re -independence requirement. Thus whether the flow in the smallest scale geometry can satisfy Re -independence is an important condition for wind-tunnel measurements to reproduce the real flow and dispersion through different scale urban areas.

However, most previous studies on the Re -independence have been focused on the external flow, such as the flow passing over a single building or building groups [21-23]. For the internal flow, such as the flow passing the indoor scale model through the open windows in this study, the concept of Re -independence of the flow structure is no more applicable. The air flow passing through the open windows and entering the rooms will be the turbulent flow in the reality but would be likely laminar flow in wind-tunnel tests. According to the above analysis, when the reduced scale model is used, it is necessary to confirm the flow-structure independence for the studied multiscale models, including the street and indoor scales.

The above discussion has set forth the background of the present study. The first aim of the present work is to construct a coupled CFD model for accurately simulating the multiscale problems on the flow and pollutant dispersion within different scale urban areas. The feasibility of the adopted turbulence models are examined by comparing their

predictions with the wind-tunnel measurements, which were reported in our previous work, where a simple multiscale model with three geometry scales (neighborhood scale – street scale – indoor scale) was constructed by considering the thermal effects with the range of Ri from 0 to 4.77 [24]. The influence of turbulent Schmidt number (Sc_t) on the pollutant dispersion is considered. The second purpose of this study is to clarify whether there exists Re-independence for the flow structures in the scaled indoor environment. The third one is to examine the effects of the outdoor environment on the [IAQ \(indoor air quality\)](#) by numerical and experimental methods.

2. Description of the experiments

The wind-tunnel experiments were completed in TJ-1 and TJ-4 wind-tunnel laboratories (The State Key Laboratory of Civil Engineering for Disaster Prevention, Tongji University, Shanghai). To model the flow and pollutant dispersion within the different scale urban areas, a simple multiscale physical model is designed with a scale ratio of 1:100 (Fig. 2). This model can be divided into three different scales: (a) neighborhood scale; (b) street scale and (c) indoor scale. Their detailed dimensions are shown in Fig. 2, where L is the characteristic height equal to 0.16 m. The pollutant source with dimensions of $2\text{ cm} \times 2\text{ cm} \times 2\text{ cm}$ is located on the rooftop of the low-rise building and SF_6 is used as a tracer gas and released from the top surface of the pollutant source with a constant release rate of 0.8 L/min. The ground between the two buildings is heated by rubber heating mats with dimensions of $L \times L$ to model the solar radiation. According to the nondimensionalization of the governing equations by Snyder [22], for an accurate modeling, several dimensionless parameters in the prototype must be duplicated in the

wind-tunnel experiments. The first important dimensionless parameter is Reynolds number (Re), which represents the ratio of inertial and viscous forces, defined as:

$$Re = \frac{U_R H}{\nu} \quad (1)$$

where U_R represents the reference velocity (m/s) which is the free-stream velocity in this study. H is the building height (m) and ν is the kinematic viscosity of the air (m²/s). The Re here is called building height Reynolds number (Re_H). To consider the effect of the buoyancy force induced by the ground heating on the flow fields, Richardson number (Ri) is used, defined as:

$$Ri = \frac{Gr}{Re^2} = \frac{g\beta(T_w - T_{ref})H^3}{\nu^2} \frac{\nu^2}{U_R^2 H^2} = \frac{g\beta(T_w - T_{ref})H}{U_R^2} \quad (2)$$

where Gr is Grashof number, T_w is the surface temperature of the heated ground (K), T_{ref} is the ambient temperature (K), g is the gravitational acceleration (m/s²) and β is the volumetric expansion coefficient (1/K).

The wind-tunnel experiments were run at the mean streamwise velocities from 0.4 to 6 m/s coupled with the temperature of the heated ground from 15 to 200 °C, which corresponded to several cases with a wide range of Ri from 0 to 4.77 (shown in Table 1). The inlet profiles at the inlet plane (Fig. 2(a)) were measured by 3-D Cobra probe with the accuracy of ± 0.1 m/s, which can be used to provide realistic inlet conditions for the numerical simulations. PIV, T-type thermocouples and SF₆ quantitative detector were used to measure flow fields, temperature and pollutant distributions [24].

3. Numerical models

3.1 Governing equation

The RANS turbulence models can provide reasonable solutions for a wide range of problems on the basic flow, heat and pollutant transfer while requiring relatively low computational cost. The generation of buoyancy force due to the thermal effect between the two building models was taken into account in this work, and the governing equations include: the continuity equation in incompressible form,

$$\frac{\partial u_j}{\partial x_j} = 0 \quad (3)$$

the RANS equation,

$$\frac{\partial(u_j u_i)}{\partial x_j} = \frac{\partial}{\partial x_j} \left((\mu + \mu_t) \frac{\partial u_i}{\partial x_j} \right) - \frac{1}{\rho} \frac{\partial p}{\partial x_i} + g_i \quad (4)$$

the energy equation,

$$\frac{\partial(u_j T)}{\partial x_j} = \frac{\partial}{\partial x_j} \left(\left(\frac{\mu}{Pr} + \frac{\mu_t}{\sigma_t} \right) \frac{\partial T}{\partial x_j} \right) + S_T \quad (5)$$

and a transport equation for a passive advection-diffusion,

$$\frac{\partial(\rho u_j c_i)}{\partial x_j} = \frac{\partial}{\partial x_j} \left(\left(\rho D_{i,m} + \frac{\mu_t}{Sc_t} \right) \frac{\partial c_i}{\partial x_j} \right) + S_i \quad (6)$$

where u_i is the time-averaged velocity (m/s); u_j is the velocity vector (m/s); μ is the molecular viscosity (Pa·s), μ_t is the turbulent viscosity (Pa·s); p is the pressure (Pa), g_i is the gravitational body force (m/s²), ρ is the density (kg/m³); T is the temperature (K); Pr is Prandtl number; σ_t is the turbulent Prandtl number (also written as $Prt = 0.85$); S_T is the heat source term (W/(m³·s)); c_i is the local mass concentration of SF₆ (kg/m³); $D_{i,m}$ is the diffusion coefficient for SF₆ (m²/s); S_i denotes the pollution source term (kg/(m³·s)); Sc_t is the turbulent Schmidt number which are adopted to consider the effect on SF₆ dispersion.

The standard k - ε model (SKE) and realizable k - ε model (RLKE) with the standard wall functions (SWFs) and the non-equilibrium wall functions (NEWFs) for the near-wall treatments were used to model turbulence flow for their good performance in modeling the flow structures [12,14]. Although the low-Reynolds number modeling (LRNM) maybe give more accurate simulation results, it requires a very fine cell close to the wall to resolve the near wall region, which increases the large difficulty of grid generation and computing time cost. Moreover, the grid used by LRNM is unsuitable for the high-Reynolds number models (SKE and RLKE) because the so fine cell close to the wall cannot satisfy the first node near the wall locating out of the viscous sub-layer [12,25]. The equations of the two adopted turbulence models are provided in Table 2.

The standard wall functions are based on the work of Launder and Spalding [26], and have been worked reasonably well for a broad range of wall-boundary flows. The law-of-the-wall for mean velocity yields,

$$U^* = \begin{cases} y^* & (y^* > 11.225) \\ \frac{1}{\kappa} \ln(Ey^*) & (y^* < 11.225) \end{cases} \quad (7)$$

where

$$U^* = \frac{U_P C_\mu k_P^{1/2}}{\tau_w / \rho} \quad (8)$$

is the dimensionless velocity.

$$y^* = \frac{\rho C_\mu^{1/4} k_P^{1/2} y_P}{\mu} \quad (9)$$

is the dimensionless distance from the wall and κ is von Karman constant ($= 0.4187$), E is the empirical constant ($= 9.793$), U_P is the mean velocity of the fluid at the near-wall node P, k_P is the turbulence kinetic energy at the near-wall node P, y_P is the distance from

the point P to the wall, μ is the dynamic viscosity of the fluid. The temperature wall functions include the contribution from the viscous heating and for the incompressible flow calculations the law-of-the-wall for the temperature field has the following composite form:

$$T^* = \frac{(T_w - T_p)(C_\mu^{1/4} k_p^{1/2})}{(q_w / \rho c_p)} = \begin{cases} Pr y^* & (y^* < y_T^*) \\ Pr_t \left[\frac{1}{\kappa} \ln(E y^*) + P \right] & (y^* > y_T^*) \end{cases} \quad (10)$$

where P is given by Jayatilleke [27]:

$$P = 9.24 \left(\frac{Pr}{Pr_t} - 1 \right) \left(\frac{Pr}{Pr_t} \right)^{-1/4} \quad (11)$$

and y_T^* is the dimensionless thermal sublayer thickness, c_p is the specific heat of the fluid, q_w the wall heat flux, T_p is the temperature at the first near-wall node P, T_w is the temperature at the wall, Pr is the molecular Prandtl number ($= \mu c_p / a$, a is the coefficient of heat diffusion), Pr_t is the turbulent Prandtl number ($= 0.85$ at the wall).

However, the standard wall functions tend to become less reliable when the flow situations depart from the ideal conditions and are subjected to severe pressure gradients and strong non-equilibrium. The non-equilibrium wall functions are introduced and can potentially improve the results in above situations [28]. The law-of-the-wall for mean temperature remains the same as in the standard wall functions described above. The log-law for mean velocity sensitized to the pressure gradients is

$$\frac{\tilde{U} C_\mu^{1/4} k^{1/2}}{\tau_w / \rho} = \frac{1}{\kappa} \left(E \frac{\rho C_\mu^{1/4} k^{1/2} y}{\mu} \right) \quad (12)$$

where

$$\tilde{U} = U - \frac{1}{2} \frac{dp}{dx} \left[\frac{y_v}{\rho \kappa \sqrt{k}} \ln \left(\frac{y}{y_v} \right) + \frac{y - y_v}{\rho \kappa \sqrt{k}} + \frac{y_v^2}{\mu} \right] \quad (13)$$

244 and y_v is the physical viscous sublayer thickness, and computed from

$$245 \quad y_v = \frac{\mu y_v^*}{\rho C_\mu^{1/4} k_p^{1/2}} \quad (14)$$

246 where $y_v^* = 11.225$.

247 In this study, Boussinesq model was used to treat the variable air density in which the
248 air density is taken as a constant in all terms of the solved equations, except for the
249 buoyancy term in the momentum equation:

$$250 \quad (\rho - \rho_0)g = -\rho_0 \beta (T - T_0)g \quad (15)$$

251 where ρ_0 is the referenced density of the air flow (kg/m^3); T_0 is the referenced temperature
252 (K); Equation (15) is obtained by the Boussinesq approximation $\rho = \rho_0 (1 - \beta \Delta T)$ to
253 replace the buoyancy terms (pressure and gravity terms). This approximation is
254 acceptable as long as changes in actual density are small. Specifically, it is valid when β
255 $(T - T_0) \ll 1$, and should not be used if the temperature difference in the domain is large.
256 According to the values of $\beta \Delta T$ shown in Table 1, Cases 2 and 3 are modeled with
257 Boussinesq approximation. For Cases 4 ~ 8, the incompressible ideal gas law was chosen
258 to define the density, thus the additional equation, Eq. (16), should be solved,

$$259 \quad \rho = \frac{M p_{op}}{R T_l} \quad (16)$$

260 where R is the universal gas constant ($\text{J}/(\text{mol} \cdot \text{K})$), M is the molecular weight of the air
261 (g/mol), p_{op} is the operating pressure (pa), and T_l is the local temperature (K). Except for
262 density, the specific heat capacity, the thermal conductivity and the viscosity were
263 approximated as a piecewise-linear function of the temperature in the Navier-Stokes

equations. The eight studied cases and their correspondent parameters, including Ri , Re and Gr numbers, are shown in Table 1.

3.2 Computational domain and grid generation

Since the multiscale building models are symmetrical geometries along the x - z plane (Fig. 2), a half of the whole geometry was taken as the computational domain (Fig. 3(a)). Some preliminary simulations of the entire domain were carried out to explore the effects of the size of the computational domain (L_x , L_y and L_z in Fig. 24(a)), and to ensure the computational domain large enough that the calculation results are independent of the domain. The appropriate computational domain were finally determined as $L_{x-} = L_{y+} = L_{z+} = 3L$ and $L_{x+} = 5L$, where L equals to 0.16 m.

To reduce the computational cost, a non-uniform structured grid system is used to discretize the computational domain. The generated grid system can be described as follows: a grid with the finest resolution of the cell scale of 0.001 - 0.004 m is used to discretize the room scale geometry and the rooftop stack, in the street scale a finer grid resolution of 0.004 - 0.01m is used, and the farther the distance from the street, the coarser the grid is applied to discretize the neighborhood scale geometry, where the ratio of two neighborhood cell dimensions is kept from 1.02 - 1.16 m. Four grid systems with different resolution based on guidelines of Franke et al. [29] are used to examine the grid sensitivity. The grid numbers of this four systems are 6.4 E+05 (Grid-1), 8.8 E+05 (Grid-2), 1.42 E+06 (Grid-3) and 2.68 E+06 (Grid-4), respectively. The velocity distributions along the room horizontal centerline with the the same numerical setups using the four grid systems are compared in Fig. 4. Obviously, the results from Grid-3 and Grid-4 are fairly close to each other, and the relative deviation of between Grid-3 and Grid-4 is less than 4%, which indicates that the grid resolution of Grid-3 is adequate to guarantee the

computational precision. For a reliable modeling, Y^+ should be between 11.5 and 300 to ensure the accuracy for the application of high-Reynolds turbulent model [25,30]. In this study, the cases with Grid-3 system had the Y^+ values between 11 and 15.

3.3 Boundary conditions and computational solution

The measured velocities on the inlet plane are uniform and imposed as the inlet boundary conditions for the numerical model, and the inlet turbulent kinetic energy (k) and dissipation (ε) [31] are defined as,

$$k = \frac{3}{2}(\bar{u}I)^2 \quad (17)$$

where,

$$I = 0.16(Re_D)^{-\frac{1}{8}} \quad (18)$$

where $\bar{u}$ is the mean velocity (m/s), I is the turbulent intensity (%), Re_D is the Reynolds number of the inlet plane based on the hydraulic diameter of the wind-tunnel cross section. The dissipation rate ε is specified as

$$\varepsilon = c_\mu^{\frac{3}{4}} \frac{k^{\frac{3}{2}}}{l} \quad (19)$$

where l is the turbulence length scale (m), equals to $0.07L$, L is the hydraulic diameter of inlet plane ($L = 0.8$ m), $c_\mu \approx 0.0845$. The other detailed boundary conditions are summarized in Table 3. For all the simulations, the SIMPLE algorithm was adopted to handle the velocity-pressure coupling and second order upwind scheme was applied to discretize the convective term for a reasonable numerical precision [25,30]. The convergence criterion of the iteration is the relative residuals of all dependent variable being less than 10^{-5} . This is a general practice for examining convergence and well-accepted in similar simulations [32,33].

4. Results and Discussion

4.1 Validation for the CFD models

In this section, to validate the CFD models, dimensionless velocity profiles in the street and indoor scales are firstly compared with wind-tunnel measurements qualitatively and quantitatively. The streamlines and temperature distributions in the street canyon for some cases are also used for further validations. Then the effects of Sc_t on pollutant dispersion are considered to determine a better choice of CFD models. The validated CFD models then are applied to investigate the flow-structure independence in the street and indoor scale models. Finally, the effects of the outdoor environment on the IAQ are studied.

4.1.1. Velocity comparison

As indicated above, the flow studied is the mixed one for which Ri is the key parameter. Therefore velocity distributions will be presented for different Ri cases. In Fig. 5, the numerical results from four turbulence models (SKE-SWFs, SKE-NEWFs, RLKE-SWFs and RLKE-NEWFs) and wind-tunnel measurements are presented for the normalized velocity profiles along the vertical centerline of the street canyon for eight Ri cases. Some features can be observed. Firstly, from the velocity profiles of wind-tunnel measurements, the minimum velocity (marked with the dotted lines in the figures) in the vertical centerline is moving upwards with the increase of Ri , which may be ascribed to the upward movement of the vortex center due to the buoyancy effect. Secondly, when Ri is less than 0.85 (Cases 1-3), for the SKE models with two different wall functions (SKE-SWFs, SKE-NEWFs) there is a good agreement between the test data and the predicted velocity profiles. Thirdly, when Ri is equal to or larger than 0.85 (Cases 4-8), the velocity distributions from RLKE-SWFs ($Ri = 1.22$) and RLKE-NEWFs ($Ri = 1.56, 4.77$) have a

334 better agreement. Thus generally speaking for mixed turbulent flow when Richardson
 335 number is small, SKE may be used, while for high Ri case RLKE is a better choice. From
 336 Fig. 5 one thing is worth noting: Case 5 and Case 6 have the same Ri but different Gr and
 337 Re . However, either the measured or the predicted velocity profiles of the two cases are
 338 very close to each other, once again indicating that for the mixed flow studied Richardson
 339 number is the dependent criteria.

340 Come here to analyze the difference between measured and predicted velocities
 341 qualitatively. The correlation coefficient, R^2 , is adopted to investigate the relative
 342 agreement between numerical results and wind-tunnel measurements. R^2 is defined as
 343 follows,

$$344 \quad R^2 = \frac{\left(\sum (U_{ref} - \bar{U}_{ref})(U_{com} - \bar{U}_{com})\right)^2}{\sum (U_{ref} - \bar{U}_{ref})^2 \sum (U_{com} - \bar{U}_{com})^2} \quad (20)$$

345 where U_{ref} is the reference velocity (wind-tunnel results) (m/s), U_{com} is the numerically
 346 predicted velocity (m/s). Obviously for the perfect case R^2 has its upper limit of 1.0. The
 347 closer the value of U_{com} to U_{ref} , the greater value of the correlation coefficient. Figure 6
 348 compares R^2 of the dimensionless velocity distributions shown in Fig. 5. It can be seen
 349 that for Cases 1-4 ($Ri \leq 0.85$), the values of R^2 of SKE-SWFs are all close to 0.9 and
 350 larger than other models. With regard to Cases 5 and 6, the calculation results of RLKE-
 351 SWFs have a better agreement with the test data than the other models. When $Ri > 1.22$
 352 (Cases 7 and 8), the agreement of results from RLKE-NEWFs with wind-tunnel test data
 353 is a bit better. From the above quantitative analysis, it can be found that for the problem
 354 studied there is no general turbulence model which can satisfactorily simulate the
 355 different cases. Furthermore, it is worth noting that with the increase of Ri , the values of
 356 R^2 become less and less, which can be attributed to the more and more complex flow

structures caused by the interaction between the inertial force and the increasing buoyancy force generated by the heated ground.

In Fig. 7, normalized velocity distributions along the horizontal centerline in the indoor scale (red dotted line in first figure shown in Fig. 7) for different Ri cases are compared between wind-tunnel measurements and numerical simulations with different turbulence models. In general, due to the ventilation through the open windows, the indoor flow structures are mainly influenced by the flow structures in the street from the qualitative overview, the matching degree of the indoor velocity profiles between the measured data and numerical simulations is worse than that in the street scale (Fig. 5 vs. Fig. 7), which may indicate that a little difference of the flow structure in the street scale will lead to a large discrepancy for the flow structures in the indoor scale, especially if the window dimension (interface of the two geometry scales) is a small one. However, generally speaking, following feature may be noted. Up to $Ri = 1.22$, the results of SKE-SWFs have a better agreement compared with others, while when $Ri = 1.56$ and 4.77 the agreement of RLKE-NEWFs seems better. This situation is quite similar with those obtained for Fig. 5. Figure 8 shows the correlation coefficients (R^2) of the dimensionless velocities in the street scale compared with the indoor scale both calculated by using the models of SKE-SWFs and RLKE-NEWFs. From this figure, we can see that the values of R^2 for both the street scale and the indoor scale (the left column and right column, respectively for each case) are getting smaller with the increase of Ri . And it is noteworthy that for all the studied cases, the values of R^2 in the street scale are remarkably larger than that in the indoor scale. Moreover, for Cases 1-4 with the increase of Ri the discrepancy of R^2 between this two geometry scales (the left column vs. the right column) becomes larger and larger. From the above findings, it may suggest that even for

the same physical model within different geometry scales, when the air flow from the street scale (a large scale) enters into the indoor scale (a small scale) through an open window, the difference of the flow structures in the street scale simulated by two turbulence models will be amplified in the indoor scale.

To obtain a more intuitive comparison, in Fig. 9 the normalized streamlines in the vertical central plane of the street canyon by using PIV measurements are compared with four turbulence models for Cases 1, 4, 5 and 7. For Case 1, there is no buoyancy effect and the flow field is isothermal ($Ri = 0$, $Re = 6.49E+04$, $Gr = 0$). It is remarkable that there is a vortex in the street canyon by PIV measurement, and the simulated results from SKE-SWFs and SKE-NEWFs agree with the measured data much better than those of RLKE-SWFs and RLKE-NEWFs. In addition, from the PIV measurements, the vortex center moves from the bottom left to the top right, then disappears in the street vertical central plane with the increasing Ri . Numerical simulation results from four turbulence models quantitatively agree with this variation trend. When $Ri \leq 0.85$ (Cases 1 and 4), the agreements of SKE-SWFs with the PIV test results are better than those of RLKE-SWFs and RLKE-NEWFs. When $Ri > 0.85$, the numerical results of RLKE model have a better agreement with the PIV measurements. On the whole, it can be observed that when Ri is small ($Ri \leq 0.85$), the SKE model can predict the flow structures better than RLKE model; with the increasing Ri ($Ri > 0.85$), the RLKE model can perform better and better. Moreover, it is important to note that when Ri is over 0.85, for the same turbulence model equation with the increase of Ri the effects of the two near-wall functions (SWFs and NEWFs) on the flow structures become more and more significant. These observations are generally consistent with the above-mentioned comparisons. Furthermore, it is worth noting that for Case 4, there exists an obvious small vortex in the right bottom corner

from both SKE and RLKE models with the SWFs near-wall treatment, however, there is no obvious vortex in the right bottom corner for the wind-tunnel measurements, which may indicate that due to laser light reflections on the surfaces, the flow structure very close to the surfaces could not be captured easily.

4.1.2. Temperature comparison

Fluid temperature is the key parameter for the buoyancy-induced flow. Hence its distribution and variation trend are worth discussing. In Fig. 10, contour plots of the normalized air temperature (Eq. (21)) in the vertical central plane and the horizontal line from measured data and numerical simulations are given for Cases 2, 4, 5 and 7. The normalized air temperature is defined by:

$$\theta = \frac{T - T_a}{T_w - T_a} \quad (21)$$

where T represents the measured temperature (K), T_a is the ambient temperature (= 288K), and T_w is the surface temperature of the heated ground (K). Qualitatively speaking, both measured and simulated results show that the high temperature region in the street canyon extends from the lower-left corner to the upper-right corner, then becomes more or less horizontal with the increase of Ri . Quantitatively speaking, for the following situations the agreement between predicted and measured temperature contour plots in the vertical central plane can be regarded quite well or acceptable: for Case 2 ($Ri = 0.1$) the predicted results by SKE-SWFs and SKE-NEWFs; for Case 4 ($Ri = 0.85$) and Case 5 ($Ri = 1.22$) the predicted results by SKE-SWFs; and for Case 7 ($Ri = 1.56$) the predicted results by SKE-SWFs and RLKE-SWFs. Moreover, when the temperature distributions of the horizontal line (line $z = 0.04$ m, $y = 0$ m for Case 2 in Fig.10) are concentrated, the

agreement seems better and more obvious. Especially for Cases 4 and 5 the predicted results by SKE-SWFs and RLKE-SWFs agree with the measured data quite well. In addition, note that for the same turbulence model equation, with the increasing Ri the effects of the two near-wall functions (SWFs and NEWFs) on the temperature distributions become more and more significant. Another obvious feature is that with the increase of Ri ($Ri \geq 0.85$) the SKE-NEWFs and RLKE-NEWFs models overestimate the temperature distributions by about 10% ~ 40% than wind-tunnel tests. The sudden increasing temperature in the right corners of the street canyon for Cases 4 and 5 (the results from PIV measurements, SKE-SWFs and RLKE-SWFs) can prove the existence of the secondary vortices, although they are not captured by the PIV measurements. The complicated situation of agreement between measured and numerically predicted results once again show that for the mixed flow and heat transfer studied among the four turbulence models no one can be regarded predominantly good. But relatively speaking, for low and high Ri cases SKE-SWFs and RLKE-SWFs respectively behave better. Therefore, in the following discussion for mass transfer only these two models are adopted for simulation.

4.1.3. Sc_t effect on the pollutant distribution

In this section the SKE-SWFs and RLKE-SWFs turbulence models are used to investigate the pollutant dispersion for Case 1 and Case 5, respectively. Pollutant dispersion is a mass transfer process in nature, and the correspondent dimensionless criterion is Schmidt number, which approximates the ratio of molecular momentum diffusivity (kinematic viscosity) to molecular diffusivity of mass. It is worth noting that Pr_t represents the ratio of turbulent momentum diffusivity and turbulent thermal

diffusivity and its value of 0.85 was adopted in our simulation. In turbulence models the data of turbulent Schmidt number (Sc_t) reported in the previous literatures are diverse, ranging from 0.2-1.3, basically depending on flow structures (jet, boundary layer, around building, etc.) [34]. Tominaga and Stathopoulos [34] hence concluded that the value of Sc_t has a significant impact on the prediction accuracy of mass transfer, and suggested the Sc_t value should be selected with careful consideration of the flow characteristics. In previous CFD studies for diffusion problems around building and through urban areas, the values of Sc_t are quite similar (from 0.63 to 0.9) [34]. In our experiments SF_6 was used as the tracer gas. To the author's knowledge this tracer gas has not been used in the previous studies for experimental measurement of turbulent Schmidt number. Based on the above discussion, in the present simulation a series of values, 0.1, 0.3, 0.7 and 1.0, were used to examine its effect and try to find which one can match the test data better.

To ensure the comparability of the concentration distributions among the different Ri cases, the normalized concentration, K , is adopted defined by

$$K = cu_h h L / Q \quad (22)$$

where c is the measured mass concentration of SF_6 (g/m^3), u_h is the undisturbed approaching flow speed (m/s) at the low-rise build height ($h = L/2 = 0.8$ m); L is the street length (= 0.16 m) and Q is the pollutant source strength (= 0.087 g/s, 0.8 L/min). In the numerical solution of the governing equation of mass concentration, Eq. (6), in conjunction with two turbulence models the mass wall function was assumed to have the same form as for temperature, with replacing T^* in Eq. (10) by θ .

Figure 11 shows K distributions along the vertical centerline of the street canyon from the wind-tunnel tests and CFD simulations with different Sc_t values. From the

comparison between the experimental and numerical results in Figs. 11(a) and (b), two features can be noted. First, it can be observed that when the value of Sc_t is low the concentration distribution along the height is more or less uniform, and with the increase of Sc_t concentration distribution along the height becomes more and more non-uniform, with the upper-center part being the largest. Second, it is noteworthy that the optimum value of Sc_t revealed by the present study are both 0.7 for Cases 1 and 5, although the agreement degree of Case 5 is worse than that of Case 1. This finding shows that with SF_6 as the tracer gas the turbulent Schmidt number for flow and diffusion around building and street canyon is quite consistent with previous studies summarized in Table 1 of Tominaga and Stathopoulos [34].

4.2 Flow-structure independence

As mentioned above, the *Re*-independence concept generally is not applicable to indoor flow phenomena. However, it is interesting to examine whether for the indoor flow induced by flow in street canyon may exist similar phenomenon? Obviously when the wind-tunnel scaled model is used to investigate the effect of street flow on the indoor flow the importance of the existence of such *Re*-independence of indoor flow structure is the same as the *Re*-independence of flow around building and street canyon studied by wind-tunnel scaled model.

Figure 12(a) presents the normalized velocity distributions along the street vertical centerline against the varied building Reynolds number (Re_H) with the SKE-SWFs model. Several features can be obtained. Firstly, with the increasing Re_H , the variation in the velocity distributions gradually becomes mild, which implies that the difference in velocity distributions is diminished with increasing Re_H . Secondly, the local diminishing

rate of the different velocities varies with the height: in the local region ($0.05 < z/H < 0.3$), when Re_H is over $2.16E+03$ ($U_R > 0.2$ m/s) the dimensionless velocities are close to each other; while $z/H > 0.3$ or < 0.05 , it requires Re_H no less than $4.32E+03$ ($U_R \geq 0.4$ m/s) to ensure the normalized velocity distributions squeeze into a narrow region. From the above observation, it can be concluded that for the external flow in the street scale, there exists a critical Re_H ($Re_{H,crit}$), over which the flow structure can be regarded as independence on the Re_H , for which the suggested value is equal to $4.32E+03$.

Figure 12(b) shows the numerically predicted dimensionless velocity contours and streamlines in the room vertical central plane with different Re_H . It can be observed that the flow structures in the room change a lot with the increasing of Re_H when the value of Re_H is less than $7.57E+03$, however, when the value of Re_H is larger than $7.57E+03$ ($U_R \geq 0.7$ m/s) not too much effect of Re_H on the indoor flow structure can be found, and the coordinates of the vortex center in the indoor vertical central plane almost keeps unchanged. Thus for the situation studied $7.57E+03$ can be regarded as the critical Reynolds number for the Re -independence of indoor flow structure, which is appreciably larger than $Re_{H,crit}$ mentioned above ($4.32E+03$). This is perhaps because the indoor flow is affected by the relatively stronger viscous force generated by the surrounding walls. To make it meaningful for the wind-tunnel tests in both street and indoor scales, we suggest the value of $Re_{H,crit}$ for this multiscale model is $7.57E+03$, which indicates that the smallest velocity for the wind-tunnel tests is $U_R = 0.7$ m/s for this study.

4.3 The effects of outdoor environment on the IAQ

The physical phenomenon of the effects of outdoor environment on the IAQ can be described as follows: the pollutants released from the stack on the rooftop of the low-rise

building follow with the mixed flow stream coming from the neighborhood scale into the street canyon (street scale), then enter into the indoor scale of the high-rise building through the ventilation and affect the health of residents in the rooms. So the IAQ is mainly determined by the environmental conditions near the open windows in the street scale. For the problem studied in this paper the environmental conditions in the street scale are dependent on both inertial force and buoyancy force.

To investigate this effect, the normalized pollutant distributions (K) along the room horizontal centerline for the eight studied cases (in Table 1) are determined from wind-tunnel test and compared with CFD simulations (Cases 1-4 using SKE-SWFs; Cases 5-6 using RLKE-SWFs; Cases 7-8 using RLKE-NEWFs) in Fig. 13, where the same shape of symbols are adopted for both measured (larger one) and simulated results (small one). From the analysis of flow-structure independence in Section 4.2, the wind-tunnel results from Cases 1-7 can represent the realistic cases, although there exist some errors in Case 8. In addition, the turbulent models and near-wall treatments are chosen because they perform better for predicting the flow fields in the street and indoor scale models. An obvious feature of Fig. 13 is the almost uniform values of the normalized pollutant distributions for each case, which can be attributed to the good mixing for the airflow in the rooms. Secondly, it can be seen that the numerical models applied can perform well to predict the pollutant distributions in the indoor scales for the eight studied cases. Thirdly, it is noted that for Cases 5 and 6 (the same street Ri) both the experimental and numerical results agree with each other quite well, especially the predicted results of the two cases are totally coincide, which indicates that the mixed pollutant distributions in the indoor scale are mainly dependent on the street Ri . Furthermore, it is worth nothing that with the

increasing Ri the average pollutant concentrations along the horizontal centerlines in the indoor scale models first increase from Case 1 to Case 4, then decrease from Case 4 to Case 8. Therefore, in this studied cases there exists the most disadvantage case (Case 4, $Ri = 0.85$) to cause the worst IAQ. The above physical phenomenon can be explained as follows: with the increase of Ri from 0 to 0.85 (Case1 to Case 4), the vortex centers in the street vertical plane approach the street center gradually [24] to cause the air flow with higher pollutant concentrations nearby the open window (see Cases 1 and 4 in Fig.7). However, with the continuous increase of Ri (Ri over 1.22, Case 5), the buoyancy force seems more obvious and strong enough to push the whole vortex structure to move towards the top-right of the street vertical plane until the vortex disappears (see Cases 5 and 7 in Fig. 7). Meanwhile, with the increasing Ri , more and more clean air coming from the both sides of the street canyon is carried into the vortex structures in the street canyon, and dilutes the pollutant concentrations there, which is one of the main reason to cause the K vs. Ri variation in the indoor scale models.

5. Conclusion

In this study, a multiscale problem on the air flow and pollutant dispersion passing over the neighborhood scale – street scale – indoor scale models was studied by the coupled CFD models and the wind-tunnel measurements with a wide range of Ri from 0 to 4.77. Two different turbulence models (SKE and RLKE) with two near-wall treatment methods (SWFs and NEWFs) were applied in the coupled CFD models, then numerical models were used to investigate the multiscale problem further. The following conclusions can be obtained.

(1) The adopted coupled CFD models can predict the general flow structures and the

buoyancy effects. When $Ri \leq 0.85$, the standard $k-\varepsilon$ model (SKE) can better predict the flow fields and temperature distributions. When $Ri > 0.85$, the realizable $k-\varepsilon$ model (RLKE) can perform better. For the same turbulence model equation with the increase of Ri the effects of the two near-wall functions (SWFs and NEWFs) on the flow structures and temperature distributions become more and more significant.

(2) When SF_6 is used as the tracer gas the optimal value of Sc_t is 0.7 for the studied multiscale models.

(3) It is numerically found that for the indoor flow structure caused by an outer street flow there also exists a Re -independence region.

(4) For the related indoor flow of the multiscale model studied, the critical Reynolds number based on build height is $7.57E+03$ and only when the street Re_H is larger than this value, the wind-tunnel results both in the street and indoor scale models are meaningful to represent the realities.

(5) The IAQ (indoor scale) is mainly affected by the outdoor environments (street scale) through the open windows (ventilation). For the multiscale model studied with the increase of Ri the average pollutant concentrations in the indoor scale first increase and then decrease.

Acknowledgments

This work is supported by the National Science Foundation of China (Grant No. 51136004), the 12th Five-Year National Key Technology R&D Program (2012BAJ02B03), and the Fundamental Research Funds for the Central Universities. Wind-tunnel test was conducted in the TJ-1 and TJ-4 wind tunnel of the State Key Laboratory of Civil Engineering for Disaster Prevention, Tongji University.

591

592 **References**

- 593 [1] E. Wennan, B. Engquist, Multiscale modeling and computation, *Not. Am. Math. Soc.*, 50 (9) (2003)
594 1062 – 1070.
- 595 [2] Y.L. He, W.Q. Tao, Multiscale simulations of heat transfer and fluid flow problems. *J. heat transfer* 134
596 (3) (2012) 031018.
- 597 [3] J.C. Chow, J.G. Watson, P.K.K. Louie, L.W.A. Chen, D. Sin, Comparison of PM2.5 carbon
598 measurement methods in Hong Kong, China. *Environ. Pollut.* 137 (2) (2005) 334 – 344.
- 599 [4] Z. Niu, S. Wang, J. Chen, F.W. Zhang, X.Q. Chen, C. He, L.F. Lin, L.Q. Yin, L.L. Xu, Source
600 contributions to carbonaceous species in PM 2.5 and their uncertainty analysis at typical urban, peri-urban
601 and background sites in southeast China. *Environ. Pollut.* 181 (2013) 107–114.
- 602 [5] Y.J. Liu, T.T. Zhang, Q.Y. Liu, R.J. Zhang, Z.Q. Sun, M.G. Zhang, Seasonal variation of physical and
603 chemical properties in TSP, PM10 and PM2.5 at a roadside site in Beijing and their influence on
604 atmospheric visibility. *Aerosol. Air Qual. Res.* 14 (3) (2014) 954 – 969.
- 605 [6] L.J. Han, W.Q. Zhou, W.F. Li, L. Li, Impact of urbanization level on urban air quality: A case of fine
606 particles (PM2.5) in Chinese cities. *Environ. Pollut.* 194 (2014) 163 – 170.
- 607 [7] R.L. Buckley, C.H. Hunter, D.W. Werth, M.T. Whiteside, K.F. Chen, C.A. Mazzola, A case study of
608 chlorine transport and fate following a large accidental release. *Atmos. Environ.* 62 (2012) 184 – 198.
- 609 [8] M.F. Milazzo, G. Ancione, R. Lisi, C. Vianello, G. Maschio, Risk management of terrorist attacks in the
610 transport of hazardous materials using dynamic geoevents. *J. Loss Prevent. Proc.* 22 (5) (2009) 625 – 633.
- 611 [9] R.E. Britter, S.R. Hanna, Flow and dispersion in urban areas. *Annu. Rev. Fluid Mech.* 35 (1) (2013)
612 469 – 496.
- 613 [10] T. van Hooff, B. Blocken, Full-scale measurements of indoor environmental conditions and natural
614 ventilation in a large semi-enclosed stadium: Possibilities and limitations for CFD validation. *J. Wind Eng.*
615 *Ind. Aerod.* 104 (2012) 330 – 341.
- 616 [11] P. Gousseau, T. Blocken, T. Stathopoulos, G.J.F. van Heijst, CFD simulation of near-field pollutant
617 dispersion on a high-resolution grid: A case study by LES and RANS for a building group in downtown
618 Montreal. *Atmos. Environ.* 45 (2011) 428 – 438.
- 619 [12] J. Allegrini, V. Dorer, J. Carmeliet, Buoyant flows in street canyons: Validation of CFD
620 simulations with wind tunnel measurements. *Build. Environ.* 72 (2014) 63 – 74.
- 621 [13] G. M. Stavrakakis, M. K. Koukou, M. G. Vrachopoulos, N. C. Markatos, Natural cross-ventilation in
622 buildings: Building-scale experiments, numerical simulation and thermal comfort evaluation. *Energy Build.*
623 40 (9) (2008) 1666 – 1681.

- [14] T. van Hooff, B. Blocken, Coupled urban wind flow and indoor natural ventilation modelling on a high-resolution grid: A case study for the Amsterdam ArenA stadium. *Environ. Model. Soft.* 25 (1) (2010) 51 – 65.
- [15] Y. Jiang, Q. Chen, Effect of fluctuating wind direction on cross natural ventilation in buildings from large eddy simulation. *Build. Environ.* 37 (4) (2002) 379 – 386.
- [16] M.J. Cook, Y. Ji, G.R. Hunt, CFD modeling of natural ventilation: combined wind and buoyancy forces. *Int. J. Vent.* 1 (3) (2003) 169 – 180.
- [17] Q. Chen, Ventilation performance prediction for buildings: a method overview and recent applications. *Build. Environ.* 44 (4) (2009) 848 – 858.
- [18] Q.H. Nie, Y. Joshi, Multiscale thermal modeling methodology for thermoelectrically cooled electronic cabinet, *Numer. Heat Trans. A-Appl.* 53 (3) (2008) 225 – 248.
- [19] P.Y. Cui, Z. Li, W.Q. Tao, Wind tunnel experiments and multiscale modeling for effects of traffic exhausts on the indoor air quality within urban-scale regions. *Indoor Air 2014 - 13th International Conference on Indoor Air Quality and Climate*, (2014) 545 – 552.
- [20] T. van Hooff, B. Blocken. On the effect of wind direction and urban surroundings on natural ventilation of a large semi-enclosed stadium. *Comput. Fluids*, 39 (7) (2010) 1146 – 1155.
- [21] P.Y. Cui, Z. Li, W.Q. Tao, Investigation of Re-independence of turbulent flow and pollutant dispersion in urban street canyon using numerical wind tunnel (NWT) models. *Int. J. Heat Mass Transfer* 79 (2014) 176 – 188.
- [22] W.H. Snyder, Similarity criteria for the application of fluid models to the study of air pollution meteorology. *Boundary-Layer Meteorol.* 3 (1972) 113 – 134.
- [23] K. Uehara, S. Murakami, S. Oikawa, S. Wakamatsu, Wind tunnel experiments on how thermal stratification affects flow in and above urban street canyons. *Atmos. Environ.* 34 (2000) 1553 – 1562.
- [24] P.Y. Cui, Z. Li, W.Q. Tao, Wind-tunnel measurements for the thermal effects on the air flow and pollutant dispersion through different scale urban areas. *Build. Environ.* 97 (2016) 137 – 151.
- [25] W.Q. Tao, *Numerical Heat Transfer*, second ed., Xi'an Jiaotong University Press, Xi'an, 2001.
- [26] B.E. Launder, D.B. Spalding, The numerical computation of turbulent flows. *Comput. Methods Appl. Mech. Eng.* 3 (2) (1974) 269 – 289.
- [27] C. Jayatilaka, The influence of prandtl number and surface roughness on the resistance of the laminar sublayer to momentum and heat transfer. *Prog. Heat Mass Transfer* 1 (1969) 193 – 321.
- [28] S.E. Kim, D. Choudhury, A near-wall treatment using wall functions sensitized to pressure gradient. In: *ASME FED (Ed.), Symp. on Separated and Complex Flows* 217 (1995) 249 – 256.
- [29] J. Franke, A. Hellsten, H. Schlünzen, B. Carissimo, Best practice guideline for the CFD simulation of flows in the urban environment, 2007 May.
- [30] S.V. Patankar, *Numerical Heat Transfer and Fluid Flow*, CRC Press, New York, 1980.
- [31] Fluent Inc., 2006. *Fluent user's guide*.

- [32] J. Franke, A. Hellsten, K.H. Schlunzen, B. Carissimo, The cost 732 best practice guideline for CFD simulation of flows in the urban environment: a summary. *Int. J. Environ. Pollut.* 44 (1–4) (2011) 419 – 427.
- [33] S. Di Sabatino, R. Buccolieri, B. Pulvirenti, R. Britter, Simulations of pollutant dispersion within idealised urban-type geometries with CFD and integral models. *Atmos. Environ.* 41 (37) (2007), 8316 – 8329.
- [34] Y. Tominaga, T. Stathopoulos, Turbulent Schmidt numbers for CFD analysis with various types of flowfield. *Atmos. Environ.* 41 (37) (2007) 8091 – 8099.

Captions

- Table 1** Parameters for the eight studied cases.
- Table 2** The governing equations for two k - ϵ turbulence models.
- Table 3** The boundary conditions of the CFD simulations.

Table 1 Parameters for the eight studied cases.

Cases	U_R (m/s)	$\Delta T(^{\circ}\text{C})$	Gr	Re	$Ri=Gr/Re^2$	$\beta \cdot \Delta T$	Density (ρ)
1	6.0	0	0	6.49E+04	0	0	Constant
2	2.8	13	8.10E+06	3.03E+04	0.01	0.04	Boussinesq approximation
3	1.4	55.5	3.23E+07	1.51E+04	0.14	0.18	
4	0.7	88.5	4.89E+07	7.57E+03	0.85	0.27	Incompressible ideal gas law
5	0.7	135	6.97E+07	7.57E+03	1.22	0.38	
6	0.8	185	8.93E+07	8.56E+03	1.22	0.49	
7	0.7	185	8.93E+07	7.57E+03	1.56	0.49	
8	0.4	185	8.93E+07	4.32E+03	4.77	0.49	

Table 2 The governing equations for two k - ε turbulence models.

	Eddy viscosity η_t	Transport equation of k	Transport equation of ε
Standard k - ε model (SKE)	$\eta_t = \rho C_\mu \frac{k^2}{\varepsilon},$ $C_\mu = 0.09$	$\frac{\partial(\rho k)}{\partial t} + \frac{\partial(\rho k u_j)}{\partial x_j} = \frac{\partial}{\partial x_j} \left[\left(\eta + \frac{\eta_t}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right] +$ $G_k - \rho \varepsilon$ $G_k = \eta_t \frac{\partial u_i}{\partial x_j} \left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \quad \sigma_k = 1.0$	$\frac{\partial(\rho \varepsilon)}{\partial t} + \frac{\partial(\rho \varepsilon u_j)}{\partial x_j} = \frac{\partial}{\partial x_j} \left[\left(\eta + \frac{\eta_t}{\sigma_\varepsilon} \right) \frac{\partial \varepsilon}{\partial x_j} \right] +$ $\frac{\varepsilon}{k} (C_{\varepsilon 1} G_k - C_{\varepsilon 2} \rho \varepsilon)$ $C_{\varepsilon 1} = 1.44, C_{\varepsilon 2} = 1.92, \sigma_\varepsilon = 1.3$
Realizabl e k - ε model (RLKE)	$\eta_t = \rho C_\mu \frac{k^2}{\varepsilon}, C_\mu = \frac{1}{4.04 + A_s k u^* / \varepsilon}$ $u^* = \sqrt{S_{ij} S_{ij} + \tilde{\Omega}_{ij} \tilde{\Omega}_{ij}}, \tilde{\Omega}_{ij} = \Omega_{ij} - 2 \varepsilon_{ijk} \omega_k$ $A_s = \sqrt{6} \cos \phi, \phi = \frac{1}{3} \arccos(\sqrt{6} W)$ $W = \frac{S_{ij} S_{ik} S_{jk}}{\tilde{S}^3}, \tilde{S} = \sqrt{S_{ij} S_{ij}}$	$\frac{\partial(\rho k)}{\partial t} + \frac{\partial(\rho k u_j)}{\partial x_j} = \frac{\partial}{\partial x_j} \left[\left(\eta + \frac{\eta_t}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right] +$ $G_k - \varepsilon, \quad \sigma_k = 1.0$	$\frac{\partial(\rho \varepsilon)}{\partial t} + \frac{\partial(\rho \varepsilon u_j)}{\partial x_j} = \frac{\partial}{\partial x_j} \left[\left(\eta + \frac{\eta_t}{\sigma_\varepsilon} \right) \frac{\partial \varepsilon}{\partial x_j} \right] +$ $C_1 \rho S_\varepsilon - C_2 \rho \frac{\varepsilon^2}{k + \sqrt{\nu \varepsilon}}$ $C_1 = \max \left[0.43, \frac{\tilde{\eta}}{\tilde{\eta} + 5} \right], C_2 = 1.9, \sigma_\varepsilon = 1.2$

Table 3 The boundary conditions of the CFD simulations.

Boundary conditions	Velocity	Temperature	Pollutant
Inlet plane	Constant	T_a	0
Outlet plane	$du/dx = 0$	$dT/dx = 0$	$dc/dx = 0$
Center plane	$du/dy = 0$; $dw/dy = 0$; $v = 0$.	$dT/dy = 0$	$dc/dy = 0$
Top/right/left planes	$du/dy = 0$; $dw/dy = 0$; $v = 0$.	$dT/dy = 0$	$dc/dy = 0$
Bottom plane	SWFs or NEWFs	T_a	0
Street ground	SWFs or NEWFs	T	0
Pollutant source	0	T_a	Constant

Captions

Fig. 1. Multiscale phenomena of air flow and pollutant dispersion in urban areas.

Fig. 2. The configurations of multiscale models and experimental setup.

Fig. 3. Computational domain and grid generation.

Fig. 4. The grid independence analysis (velocity distribution along the horizontal centerline of the room).

Fig. 5. Normalized velocity distributions along the vertical centerline of the street canyon for different Ri cases. Wind tunnel measurements and CFD simulations by using four turbulence models (SKE-SWFs, SKE-NEWFs, RLKE-SWFs and RLKE-NEWFs).

Fig. 6. Correlation coefficients (R^2) between wind tunnel measurements and different numerical models along the vertical center line in the street scale.

Fig. 7. Normalized velocity distributions along the horizontal centerline in the indoor scale for different Ri cases. Wind tunnel measurements and CFD simulations by using four turbulence models (SKE-SWFs, SKE-NEWFs, RLKE-SWFs and RLKE-NEWFs).

Fig. 8. Correlation coefficients (R^2) among different numerical models compared between the street scale and indoor scale.

Fig. 9. Normalized streamlines in the vertical central plane of street canyon. Wind tunnel measurements and CFD simulations by using four turbulence models (SKE-SWFs, SKE-NEWFs, RLKE-SWFs and RLKE-NEWFs).

Fig. 10. Distributions of dimensionless temperature (θ) in the vertical central plane of street canyon for different cases. Wind tunnel measurements and CFD simulations by using four turbulence models (SKE-SWFs, SKE-NEWFs, RLKE-SWFs and RLKE-NEWFs).

Fig. 11. Normalized concentration distributions along the vertical centerline of the street canyon. Wind tunnel measurements and CFD simulations by using two turbulence models (SKE-NEWFs and RLKE-SWFs) with different Sc_t numbers.

Fig. 12. Flow structure independence analysis for the street scale and indoor scale models.

Fig. 13. Normalized pollutant distributions along the room horizontal center lines for different cases. Wind tunnel measurements (large symbols) and CFD simulations (small symbols).

Fig. 1. Multiscale phenomena of air flow and pollutant dispersion in urban areas.

Fig. 2. The configurations of multiscale models and experimental setup.

(a) Computational domain

(b) Grid display on x - z plane

Fig. 3. Computational domain and grid generation.

Fig. 4. The grid independence analysis (velocity distribution along the horizontal centerline of the room).

Case 1: $Ri = 0$, $Re = 6.49E+04$, $Gr = 0$

Case 2: $Ri = 0.01$, $Re = 3.03E+04$, $Gr = 8.10E+06$

Case 3: $Ri = 0.14$, $Re = 1.51E+04$, $Gr = 3.23E+07$

Case 4: $Ri = 0.85$, $Re = 7.57E+03$, $Gr = 4.89E+07$

Case 5: $Ri = 1.22$, $Re = 7.57E+03$, $Gr = 6.97E+07$

Case 6: $Ri = 1.22$, $Re = 8.56E+03$, $Gr = 8.93E+07$

Case 7: $Ri = 1.56$, $Re = 7.57E+03$, $Gr = 8.93E+07$ Case 8: $Ri = 4.77$, $Re = 4.32E+03$, $Gr = 8.93E+07$

Fig. 5. Normalized velocity distributions along the vertical centerline of the street canyon for different Ri cases. Wind tunnel measurements and CFD simulations by using four turbulence models (SKE-SWFs, SKE-NEWFs, RLKE-SWFs and RLKE-NEWFs).

Fig. 6. Correlation coefficients (R^2) between wind tunnel measurements and different numerical models along the vertical centerline in the street scale.

Case 1: $Ri = 0$, $Re = 6.49E+04$, $Gr = 0$

Case 2: $Ri = 0.01$, $Re = 3.03E+04$, $Gr = 8.10E+06$

Case 3: $Ri = 0.14$, $Re = 1.51E+04$, $Gr = 3.23E+07$

Case 4: $Ri = 0.85$, $Re = 7.57E+03$, $Gr = 4.89E+07$

Case 5: $Ri = 1.22$, $Re = 7.57E+03$, $Gr = 6.97E+07$

Case 6: $Ri = 1.22$, $Re = 8.56E+03$, $Gr = 8.93E+07$

Case 7: $Ri = 1.56$, $Re = 7.57E+03$, $Gr = 8.93E+07$

Case 8: $Ri = 4.77$, $Re = 4.32E+03$, $Gr = 8.93E+07$

Fig. 7. Normalized velocity distributions along the horizontal center line in the indoor scale for different Ri cases. Wind tunnel measurements and CFD simulations by using four turbulence models (SKE-SWFs, SKE-NEWFs, RLKE-SWFs and RLKE-NEWFs).

Fig. 8. Correlation coefficients (R^2) among different numerical models compared between the street scale and indoor scale.

PIV measurement

SKE-SWFs

SKE-NEWFs

RLKE-SWFs

RLKE-NEWFs

Case 1: $Ri = 0$, $Re = 6.49E+04$, $Gr = 0$

PIV measurement

SKE-SWFs

SKE-NEWFs

RLKE-SWFs

RLKE-NEWFs

Case 4: $Ri = 0.85$, $Re = 7.57E+03$, $Gr = 4.89E+07$

PIV measurement

SKE-SWFs

SKE-NEWFs

RLKE-SWFs

RLKE-NEWFs

Case 5: $Ri = 1.22$, $Re = 7.57E+03$, $Gr = 6.97E+07$

Case 7: $Ri = 1.56$, $Re = 7.57E+03$, $Gr = 8.93E+07$

Fig. 9. Normalized streamlines in the vertical central plane of street canyon for different cases. Wind tunnel measurements and CFD simulations by using four turbulence models (SKE-SWFs, SKE-NEWFs, RLKE-SWFs and RLKE-NEWFs).

PIV measurement

SKE-SWFs

SKE-NEWFs

RLKE-SWFs

RLKE-NEWFs

θ along line $z = 0.04\text{m}$

Case 2: $Ri = 0.01$, $Re = 3.03\text{E}+04$, $Gr = 8.10\text{E}+06$

PIV measurement

SKE-SWFs

SKE-NEWFs

RLKE-SWFs

RLKE-NEWFs

θ along line $z = 0.04\text{m}$

Case 4: $Ri = 0.85$, $Re = 7.57\text{E}+03$, $Gr = 4.89\text{E}+07$

PIV measurement

SKE-SWFs

SKE-NEWFs

RLKE-SWFs

RLKE-NEWFs

θ along line $z = 0.04\text{m}$

Case 5: $Ri = 1.22$, $Re = 7.57\text{E}+03$, $Gr = 6.97\text{E}+07$

Case 7: $Ri = 1.56$, $Re = 7.57\text{E}+03$, $Gr = 8.93\text{E}+07$

Fig. 10. Distributions of dimensionless temperature (θ) in the vertical central plane of street canyon for different cases. Wind tunnel measurements and CFD simulations by using four turbulence models (SKE-SWFs, SKE-NEWFs, RLKE-SWFs and RLKE-NEWFs).

(a) Case 1: SKE-SWFs vs. EXP.

(b) Case 5: RLKE-SWFs vs. EXP.

Fig. 11. Normalized concentration distributions along the vertical centerline of the street canyon. Wind tunnel measurements and CFD simulations by using two turbulence models (SKE-SWFs and RLKE-SWFs) with different Sc_t numbers.

(a) Dimensionless velocity distributions along the vertical centerline of the street canyon with different street Re_H .

(b) Dimensionless velocity contours and streamlines in the room vertical center plane with different street Re_H .

Fig. 12. Flow structure independence analysis for the street scale and indoor scale models.

Fig. 13. Normalized pollutant distributions along the room horizontal center lines for different cases. Wind tunnel measurements (large symbols) and CFD simulations (small symbols).