Numerical Simulation of Wind Flow and Pollution Transport in Urban Street Canyons

Trong Nhan Nguyen¹, Thanh Chuyen Nguyen² and Van Thinh Nguyen²*

¹School of Architecture, University of Detroit Mercy, Detroit, MI 48221, USA
²Department of Civil and Environmental Engineering, Seoul National University, Seoul 151-744, South Korea
¹nguyentn@udmercy.edu, ²{thanh, vnguyen}@snu.ac.kr
*Corresponding Author

Abstract. In this study, a three-dimensional numerical simulation of the wind flow and pollutant transport in street canyons is carried out by a new specific solver, which is developed and coupled into an open source CFD tool box, the OpenFOAM package. The new model has been applied to study the effect of roof shapes and building heights on the flow patterns and pollutant transports in the street canyons. The numerical results have been validated against laboratory experiments. The comparisons between the simulations and observations show a good agreement on the flow patterns and pollutant transports.

Keywords: Street canyons, numerical simulation, flow pattern, pollution transport.

1 Introduction

Pollution from industrial activities, vehicle exhaust, heating and cooling systems, etc. can cause fatal harms to humans in urban street canyons, therefore investigation of flow characteristics and pollution transports in street canyons is a crucial task in the urban environment. The most important characteristics of the flow in street canyons are the wind-induced flow patterns characterizing by internal flows, flow separation and reattachment which effect on the local air quality and consequently on the human health in a city. For such flows it is very difficult to accurately calculate flow patterns and pollutant transports. The study on wind flow and pollutant transport inside and over urban street canyons have attracted great concern during last three decades due to increasing urban pollutants. Field measurements and laboratory-scale physical modeling are not only very expensive but also difficult, and somehow impossible due to the temporal and spatial scales. Inheritance from the increasing of computer technology (HPC facility), Computational Fluid Dynamics (CFD) becomes the most powerful tool for the simulation of wind flows and pollutant transport in urban street canyons. Many authors have applied CFD tools for this problem, such as Johnson and Hunter [3], Baik and Kim [1], Chan et al. [2], Li et al. [4], Yassin [8], etc. They have mainly applied Reynold Averaged Navier-Stokes (RANS) equations with k-ε turbulence closure model and its variants (RNG k-ε, realizable k-ε). Recently, Moonen et al [5] applied Large Eddy Simulation (LES) for the modeling of dispersion.
in urban street canyon. However, most of authors have used a commercial CFD software, such as CFX, Fluent (of ANSYS).

In this study, based on an open source CFD package OpenFOAM we developed a new specific solver, and coupled it into the OpenFOAM package (http://www.openfoam.com/). The OpenFOAM package is a general CFD tool box for applications in continuum mechanics problem. It is written in C++ and designed as a structure of Partial Differential Equation’s solvers which can give a professional user an opportunity to build their own specific solvers, then immerse it into the package. Based on this advantage, we develop a new solver combing the aerodynamic problem with a transport process together to facilitate the pollution transport simulation driven by the turbulent flows. In the standard library of OpenFOAM, the numerical solution is designated only for a passive scalar transport, i.e. the concentration field is solved for a given stationary velocity field, and it can deal with only constant diffusion coefficient so that it cannot take in account the effect of turbulent flows. In the new solver, the scalar transport equation is solved together with RANS (Reynolds Averaged Navier-Stokes) equations with k-ω turbulence closure model. At each time step, after updating the aerodynamic parameters, the advection-diffusion equation can be solved. A difference between the original and customized solvers is shown in Figure 1 below.

Fig. 1. A comparison between the original and customized solvers

2 Governing equations

2.1 Aerodynamic equations

For aerodynamic calculation, the RANS equations were modeled by two-equation turbulence model (k-ω). It is described as follows:
where:

\[ \sigma_k = 2.0, \quad \sigma_\omega = 2.0, \quad \gamma_1 = 0.553, \quad \beta_1 = 0.075, \quad \beta^* = 0.09 \]

It is well-known that the \( k-\omega \) model performs better predictions for the flow with strong separation and the attachment length in comparison with the \( k-\varepsilon \) model, this is a reason why we chose the \( k-\omega \) model instead of the \( k-\varepsilon \) model.

### 2.2 Scalar Transport Equation

Similarly as the same manner to obtain RANS Equations, the time-averaged transport equation for a scalar \( C \) is obtained:

\[
\frac{\partial C}{\partial t} + U_j \frac{\partial C}{\partial x_j} = \frac{\partial}{\partial x_j} \left( D \frac{\partial C}{\partial x_j} \right) - \frac{\partial}{\partial x_j} \left( \bar{w}' \bar{C}' \right)
\]

where the prime denotes a fluctuating value.

The simplest model for turbulent scalar fluxes follows from the standard gradient-diffusion hypothesis (SGHD), where the turbulent scalar flux is assumed proportional to mean scalar gradient as follows:

\[ \bar{w}' \bar{C}' = -D \frac{\partial C}{\partial x_i} \frac{\partial s}{\partial x_i} \]

So we can rewrite the equation (5) as

\[
\frac{\partial C}{\partial t} + U_j \frac{\partial C}{\partial x_j} = \frac{\partial}{\partial x_j} \left( D \frac{\partial C}{\partial x_j} \right) + D \frac{\partial C}{\partial x_i} \frac{\partial s}{\partial x_i}
\]

where \( D_i \) is turbulent diffusion coefficient which is assumed to be isotropic and homogeneous.

In equation (6), the molecular diffusion, \( D \) can be neglected because the turbulent diffusion \( D_i \) is much larger than molecular diffusion. Therefore, it can be reduced as:
The turbulent diffusion coefficient $D_t$ in equation (7) is modeled by the relationship eddy-viscosity and the turbulent Schmidt number, $S_{ct} = \frac{D_t}{\nu}$. 

In this study, a value of $S_{ct} = 0.9$ has been chosen as in previous studies of pollutant spreading in street canyons.

### 3 Boundary Conditions

At the inlet, the Dirichlet boundary condition is applied for velocity, turbulent kinetic energy and specific dissipation rate. The inlet velocity profile was prescribed in the power law as in Rafailidis [7]:

$$\frac{U(z)}{U(0)} = \left(\frac{z}{D}\right)^{\alpha}$$

The parameters are obtained from the measurement, where: $\alpha=0.28$, $D=0.002\text{m}$ is the displacement height above the ground, $U(\delta)=5\text{m/s}$ is free-stream velocity, $\delta=0.5\text{m}$ is the thickness of boundary layer.

The turbulence parameters of $k$-$\omega$ turbulence model is set as follows:

$$k = \frac{1}{2} (UI)^2, \quad \omega = C_\mu^{0.25} \frac{\nu\beta}{l}$$

where $U$ is mean value of velocity at the inlet; $I$ is the turbulent intensity; $C_\mu = 0.09$ is a turbulence model constant; $l$ is the turbulent characteristic length.

No-slip condition was set for velocity at the building walls and road surface. The turbulence parameters such as turbulent kinetic energy and specific dissipation rate were set to wall functions at the wall.

At the symmetry and outlet all variables, such as velocity, pressure and turbulence parameters were set to zero gradient boundary condition.

### 4 Validations

The numerical simulation has been validated against the experiments of Rafailidis and Schatzmann [3] and Rafailidis [4] with different flat and slanted roof shapes.

#### 4.1 Flat-shaped roof

The modeled street canyon consisted of eight idealized streets. The aspect ratio of the building height $H$ and the width $b$ of the street is 1, whereby $H=b=60\text{ mm}$. There is a steady source $C=150\text{ ppm}$ at the bottom center of the fourth street canyon. The geometry is shown in Figure 2 below

Figure 3 shows the streamline of velocity from the numerical results. It shows that the vortices are located in the center of the street canyon. Figure 4 shows a comparison between the observation and numerical results of the vertical velocity.
profile at upstream ($x/b = -0.25$), center line ($x/b = 0$) and downstream ($x/b = 0.25$) of the center line of the fourth street canyon. Figure 5 shows a comparison of the dimensionless concentration $K$ (Eq. 8) between the observation and numerical results.

**Fig. 2.** The geometry of flat-shaped roof

**Fig. 3.** Streamline of wind flow over flat-shaped roofs

**Fig. 4.** A comparison between observation and numerical results of vertical velocity profile in the center canyon (windward: $x/b=-0.25$, center: $x/b=0$, leeward: $x/b=0.25$)
Fig. 5. A comparison of dimensionless concentration \( K \) between observation and numerical results in the center canyon.

Non-dimensional concentration:

\[
K = \frac{C U H L}{Q}
\]  

(8)

Where: \( C \) [ppm] is the tracer concentration; \( U \) [m/s] is free stream velocity; \( H \) [m] is building height; \( L \) [m] is length of line source; and \( Q \) [m\(^3\)/s] is total emission strength.

4.2 Slanted-shaped roof

Similar to the flat roof case, there are 8 buildings with different slanted roof heights with the aspect ratios \( \frac{Z_H}{H} \) of 0.5, 0.33, 0.17 (\( Z_H \) is the height of the roof). There is also a steady source \( L=150 \) [ppm] at the center of the fourth street canyon. The geometry of the slanted roof shape is shown in Figure 6.

Fig. 6. The geometry of slanted-shaped roof
Fig. 7. Streamline of wind flow over slanted-shaped roofs $Z/H = 0.5$

Fig. 8. Streamline of wind flow over slanted-shaped roofs $Z/H = 0.33$

Fig. 9. Streamline of wind flow over slanted-shaped roofs $Z/H = 0.17$

Fig. 10. A comparison of dimensionless concentration $K$ between exp. and comp. at the center canyon
Figures 7, 8 and 9 show the streamline of velocity from the numerical results. It shows that the vortices of wind flow in the street canyon are lifted upward once the slope of the slanted roof is increased, which shows the same results obtained from the simulation by Yassin [8]. The location of the vortex center in the street canyon obtained from our simulation is also very similar to the numerical results of Yassin [8]. Figure 10 shows a comparison of the dimensionless concentration $K$ at upstream ($x/b = -0.25$), and downstream ($x/b = 0.25$) of the center line of the fourth street canyon.

5 Conclusion

As shown above, the numerical model has been verified against benchmarking experiments of Rafailidis and Schatzmann [6], and Rafailidis [7]. The comparisons show a reasonable agreement between the observations and numerical results. It shows the ability of the numerical model, which can be used to calculate the turbulent flows and pollutant transports in urban street canyons. However, before this model can be applied into a real-world applications, a further development and calibrations on different roof’s shape and height are of course needed. To capture a better flow patterns and pollutant tracers in the urban street canyons, we plan to implement a higher accurate turbulent calculation, such as Reynolds Stress Model (RSM) or Large Eddy Simulation (LES).

References