Journal of Marine Science and Technology

Volume 21 | Issue 2 Article 2

NUMERICAL SIMULATIONS AND WIND TUNNEL STUDIES OF POLLUTANT DISPERSION IN THE URBAN STREET CANYONS WITH DIFFERENT HEIGHT ARRANGEMENTS

Cheng-Hsin Chang
Department of Civil Engineering, Tamkang University, Tamsui, Taipei County, Taiwan, R.O.C,
cc527330s@mail.tku.edu.tw

Jin-Shian Lin
Cheetah Industrial Aero-Dyna. Tech Co., Ltd., Taipei County, Taiwan, R.O.C

Chii-Ming Cheng
Department of Civil Engineering, Tamkang University, Tamsui, Taipei County, Taiwan, R.O.C

Yung-Shan Hong
Department of Civil Engineering, Tamkang University, Tamsui, Taipei County, Taiwan, R.O.C

Recommended Citation
Chang, Cheng-Hsin; Lin, Jin-Shian; Cheng, Chii-Ming; and Hong, Yung-Shan (2013) "NUMERICAL SIMULATIONS AND WIND TUNNEL STUDIES OF POLLUTANT DISPERSION IN THE URBAN STREET CANYONS WITH DIFFERENT HEIGHT ARRANGEMENTS," *Journal of Marine Science and Technology*: Vol. 21 : Iss. 2 , Article 2.
DOI: 10.6119/JMST-012-0109-2
Available at: https://jmstt.ntou.edu.tw/journal/vol21/iss2/2

DOI: 10.6119/JMST-012-0109-2

NUMERICAL SIMULATIONS AND WIND TUNNEL STUDIES OF POLLUTANT DISPERSION IN THE URBAN STREET CANYONS WITH DIFFERENT HEIGHT ARRANGEMENTS

Cheng-Hsin Chang[1], Jin-Shian Lin[2], Chii-Ming Cheng[1], and Yung-Shan Hong[1]

Key words: wind tunnel experiment, urban street canyon, numerical simulation, dispersion.

ABSTRACT

Air pollution in big city areas resulting from exhaust emissions is a major urban problem. Often traffic pollution excess controls air pollution management decisions. There are a number of elaborate predictive models of pollutant dispersion and diffusion that address the effects of variable shapes of city buildings on pollutant concentrations, but few are fully validated. This study presents ventilation behavior in different street canyon configurations. To evaluate dispersion in a model urban street canyon, a series of tests with various street canyons with different height in upwind and downwind of street canyon are presented. These buildings were arranged in 2-D configurations with different height in upwind and downwind of street canyon. The results showed that a higher concentration of pollutants accumulates under the leeward of the street canyon due to the occurrence of a clockwise vortex inside the street canyon when the street canyon aspect ratio (B/H) is 2. On the contrary, over the windward of the street canyon, a lower concentration of pollutants accumulates due to the occurrence of an anti-clockwise vortex. The flow and dispersion of gases emitted by a line source located between two buildings inside of the urban street canyons were also determined by numerical model. Calculations were compared against CFD prediction in an Environmental Wind Tunnel of Wind Engineering Center at Tamkang University.

I. INTRODUCTION

The flow patterns that develop around individual buildings govern the wind forces on the building and the distribution pressure about the building and pollution about the building and in its wake. The superposition and interaction of flow patterns associated with adjacent buildings govern the final distribution of facade pressures and the movement of pollutants in urban and industrial complexes. Street canyon depth and width, intersection locations, canyon orientation to dominate wind directions and building geometries will determine peak pollution incidents.

Advanced technology makes computers faster and more powerful, which allows computational fluid dynamics (CFD) procedures to be applied to many experimental flow problems. Today, increasing applications of CFD to wind engineering problems include wind load of building and pollutant dispersion phenomena. Several previous studies have compared measurements made during physical modeling with numerical predictions. He and Song [8] simulated the wind flow around the Texas Tech University (TTU) building and roof corner vortex by using a Large Eddy Simulation (LES) code. They claim that the three-dimensional roof corner vortex pattern was successfully simulated and that mean values of pressure predicted were in good agreement with wind tunnel and field test measurements. Murakami *et al*. [15] generated velocity fluctuations for an inflow boundary condition for LES with prescribed spatial correlation distributions and turbulence intensity levels. To generate velocity fluctuations for an inflow boundary condition for LES is one of the most important unresolved problems in CFD research. Lee *et al*. [18] solved the LES of wind effects on bluff bodies using the finite element method, and they compared simulated results with numerical and experimental studies reported by other researchers. In this study related to air pollutants transportation in street canyons, Meroney [16] carried out wind tunnel tests with the same building heights, but with different street widths. It is important to find out how these pollutants distribution in the streets and the pollution at pedestrian level can be decreased. Configurations like these have been investigated and simulated with computational fluid dynamic software by Murakami [21], Zhang *et al*. [25], Hwang *et al*. [9] and Tsuchiya *et al*. [23].

Paper submitted 12/02/10; revised 12/08/11; accepted 01/09/12. Author for correspondence: Cheng-Hsin Chang (e-mail: cc527330s@mail.tku.edu.tw).
[1] *Department of Civil Engineering, Tamkang University, Tamsui, Taipei County, Taiwan, R.O.C.*
[2] *Cheetah Industrial Aero-Dyna. Tech Co., Ltd., Taipei County, Taiwan, R.O.C.*

Field measurement include Carpenter [1], Johnson *et al*. [10], De Paul and Shien [7], Kukkonen [14] and Tsai [22]. A limitation of direct field measurement of atmospheric phenomena is that all possible parameters are simultaneously operative. So, it is not simple to determine which are important or which are insignificant. Analytical methods have been published and based on simplified the flow models of the dispersion parameterizations by Johnson *et al*. [11], Yamartino and Wiegand [24], Lee and Park [17], and Kastner-Klein *et al*. [12].

Such phenomenon also causes lower pollutant concentrations around the rooftops of buildings in the downwind site inside the street canyon. On the contrary, in a rural area, the vortex in the street canyon becomes less stabilized, but with better ventilation. This may cause higher concentrations around the rooftops in the downwind site inside the street canyon. Kastner [13] conducted two-dimensional and three dimensional wind tunnel tests separately. In the two dimensional street canyon, one or two models with the same heights were placed in the upwind area. The results represented that more obstacles in the street canyon would cause a stabilized vortex but poor ventilation. This also caused higher concentrations at both leeward and windward sides of the street canyon.

Leitl [19] extends Meroney [16] and also conducted wind tunnel experiments with triangle rooftops added onto the models at both the leeward and windward sides. Leitl concluded that if triangle rooftops were added at two sides of the street canyon, the average concentration was the highest; the lowest concentration happened when the rooftops were added at the head of the street canyon. Using FLUENT software in a simulation of a two dimensional street canyon, the concentration of the leeward side was 58% higher than in the laboratory. Furthermore, results of the numerical simulation showed that there were not significant changes of the velocity field between the rooftop-added and non-rooftop added street canyons. However, in the numerical simulation of the three dimensional street canyon, the concentration at the leeward side had a 90% difference when compared to the results in the laboratory. But the concentration from the numerical simulation at the windward side was higher than the results with the wind tunnel test. The results were unfavorable.

Chan *et al*. [2] compared Meroney's study results with the results derived from applying FLUENT. By simulating in three turbulent models (κ-ε), the results showed that when the wind speed reaches 1 m/sec, the results from the standard κ-ε turbulent model were favorable. The statistics (parameters) from the RNG (renormalization group) κ-ε and (realization) κ-ε turbulent model were much different than the statistics from the physical experiment. Meanwhile, when the wind speed was under 0.5 m/sec, the results from the standard turbulent model and the RNG (renormalization groups) κ-ε were similar; as for the (realizable) κ-ε, the statistics were higher than the parameters obtained from the wind tunnel test. When the wind speed was greater than 2 m/sec, the statistics from the turbulent model matched the results from the wind tunnel experiment. Basically, with a slow wind speed, the experiment is affected by certain factors. And the results from the simulation can't be completely illustrated by only one turbulent model.

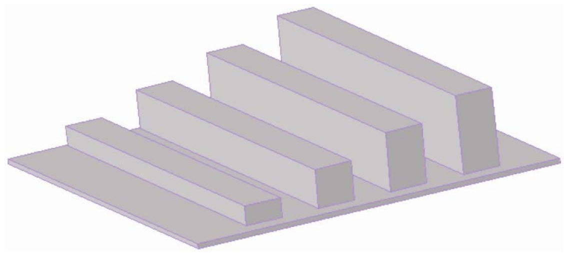

Fig. 1. Street canyon physical geometry.

II. WIND TUNNEL EXPERIMENT AND NUMERICAL MODELING

1. Wind Tunnel Modeling

Experiments were done in a low-speed environmental wind tunnel located at Tamkang University in Tamsui, Taiwan. The test section of the wind tunnel has dimensions of 3.5 × 18 × 2.0 m (width × length × height) and can generate an average wind speed of up to 16 m/sec. To avoid having the walls of the tunnel interfere with the flow rates and streamlines, the cross-sectional area of an obstacle should be less than 5% of the tunnel cross-sectional area. The street canyon model utilized in this study have a cross-sectional area approximately 4% of the 14-m^2 tunnel cross section. The boundary layer thickness from the tunnel walls showed less than a 5% change (from 14 to 14.5 cm thick) with the model present. Therefore, we can safely assume for our experiments that the tunnel wall has a negligible effect on the bulk of the flow field within the test section.

The size of the street canyon in the study was 75 cm long × 8 cm width. The heights from left to right were 4, 8, 12, 16 cm, saw Fig. 1.

2. Tracer Gas Selection and Detection

For accurate, sensitive, and consistent detection of exhaust concentrations downstream of the street canyon models, a line sources of tracer gas was released in the centerline of the street canyons. According to previous experiments done by other researchers, we expected at least 3 orders of magnitude range in tracer gas concentrations throughout the test section. Ethene was chosen as the tracer gas because it has a low background concentration (typically in the 0- to 20-parts per billion [ppb] range at Tamsui city), and is readily and sensitively quantified via GC. Ethene also has a slightly lower molecular weight (28) than the ambient air and thus might somewhat simulate the buoyancy of the hot tailpipe exhaust. In the experimental setup, we aimed for minimum tracer concentrations downstream in the parts per million (ppm)

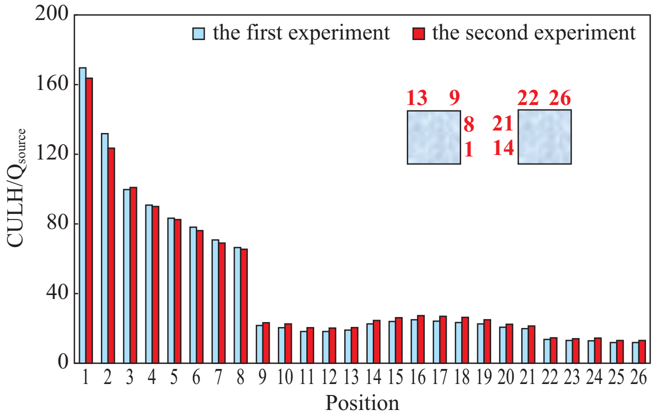

Fig. 2. Two tests on pollutant concentrations in a street canyon with the same pollutant emission (Qsource = Qair (135 l/h) + Qpropane (4 l/h)) and reference velocity (3 m/sec). (H: canyon height = 8 cm, L: canyon length = 75 cm, C: concentrations).

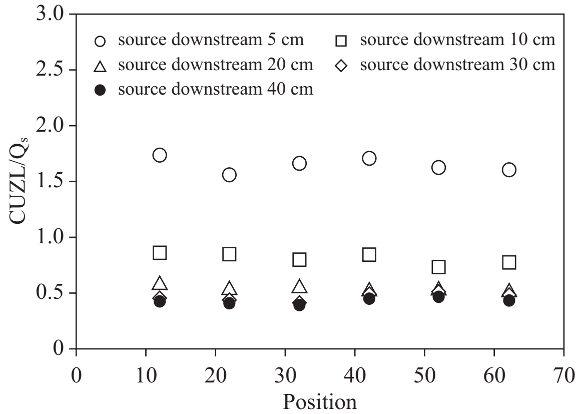

Fig. 3. Horizontal downstream dispersion of the gas plume from line source. (C: concentrations, Z: roughness length = 0.6 cm, U: reference velocity = 2 m/sec, L: canyon length = 75 cm, Qs: total pollutant emission Qsource = Qair (200 l/h) + Qpropane (4 l/h)).

range, at least 50 times higher than the ambient background level. By using ethene as our tracer compound, we do not have to worry about quantifying the effects of coagulation and/or chemical reactions on the measurements. In addition, the submicron-sized particles that are characteristic of vehicle exhaust should follow the fluid mechanical streamlines much like gas molecules. The concentration of tracer was determined by a gas chromatograph equipped with a flame-ionization detector (FID). The column was specially designed for gaseous species in the C1–C5 group. (Model: Hewlett Packard 5890 with HP-Plot Q, and DB-5msc)

In order to simulate the emission of the linear pollution source, it is important to consider the design of the linear pollution source. In Meroney's study, the deviation (error) of the test was not over 10%, and the results even surpassed Munchow's study [20]. Therefore, the results were significant and valuable.

The concentration of the sample collected from the sampler is not interfered by other factors. It also does not affect the accuracy of the sampling analysis run by gas chromatography. Before the wind tunnel test, in addition to the uniformity test, two tests were conducted in a symmetric street canyon under the same flow and wind speed (3 m/sec). Fig. 2 showed the results from the two tests. The deviation (error) of the dimensionless concentration was not over 5%, which ensured the accuracy and stability of the two tests.

Fig. 3 represented the results from the linear pollution source uniform test. Under the same level, the changes of the range of the dimensionless concentration were visible. The changes of the dimensionless concentration mainly happened 5 cm under the pollution source. But the deviation (error) was not over 8%. In short, the longer the distance from the pollution source, the smaller the change of the dimensionless concentration. Therefore, we can also conclude that the test results matched Meroney's study [16].

In fact, one-dimensional hot-film probe (IFA300) be used for measuring incident flow with different heights, variation of wind speed and turbulent concentration. According to Chang [3], Chang *et al.* [4], and Chang and Meroney [5, 6], using regression analysis, the statistics from measuring incident flow can be read into the FLUENT software. The results showed that the characteristics of incident flow matched the numerical simulation's boundary condition.

III. NUMERICAL MODELING

The numerical simulation tool used in this study was computational fluid dynamics, commercial code, Fluent. The Fluent CFD software was based on a finite volume discretization of the equations of motion. The program allows the user to specify up to 20 separate chemical reactions (either heterogeneous or homogeneous in nature), solve for temperatures, radiation, combustion, and particle or spray combustion, etc. For this study, the steady $\kappa - \varepsilon$ turbulence model and the mixture model of multiphase flow technique are adopted to calculate the pollutant dispersion in urban area.

1. Mathematical Model

The mixture model uses a single-fluid approach and the continuity equation for the mixture, the momentum equation for the mixture, the energy equation for the mixture, and the volume fraction equation for the secondary phases, as well as algebraic expressions for the relative velocities (if the phases are moving at different velocities). RANS equations govern the fluid motion subject to the continuity constraint:

The continuity equation for the mixture is

$$\frac{\partial}{\partial t}(\rho_m) + \nabla \cdot (\rho_m \vec{\upsilon}_m) = 0 \qquad (1)$$

where $\vec{\upsilon}_m$ is the mass-averaged velocity:

$$\vec{\upsilon}_m = \frac{\sum_{k=1}^{n} \alpha_k \rho_k \vec{\upsilon}_k}{\rho_m} \tag{2}$$

and ρ_m is the mixture density:

$$\rho_m = \sum_{k=1}^{n} \alpha_k \rho_k \tag{3}$$

α_k is the volume fraction of phase k.

The momentum equation for the mixture can be obtained by summing the individual momentum equations for all phases. It can be expressed as

$$\frac{\partial}{\partial t}(\rho_m \vec{\upsilon}_m) + \nabla \cdot (\rho_m \vec{\upsilon}_m \vec{\upsilon}_m) = -\nabla p + \nabla \cdot \left[\mu_m \left(\nabla \vec{\upsilon}_m + \nabla \vec{\upsilon}_m^T \right) \right]$$
$$+ \rho_m \vec{g} + \vec{F} + \nabla \cdot \left(\sum_{k=1}^{n} \alpha_k \rho_k \vec{\upsilon}_{dr,k} \vec{\upsilon}_{dr,k} \right) \tag{4}$$

Where n is the number of phases, $\vec{F}$ is a body force, and μ_m is the viscosity of the mixture:

$$\mu_m = \sum_{k=1}^{n} \alpha_k \mu_k \tag{5}$$

$\vec{\upsilon}_{dr,k}$ is the drift velocity for secondary phase k:

$$\vec{\upsilon}_{dr,k} = \vec{\upsilon}_k - \vec{\upsilon}_m \tag{6}$$

A k-ε model with a standard wall law is used for the turbulence. The turbulent kinetic energy, κ, and its dissipation rate ε, are obtained from:

$$\frac{\partial k}{\partial t} + u_j \frac{\partial k}{\partial x_j} = \frac{1}{\rho} \frac{\partial}{\partial x_j} \left[(\mu + \frac{\mu_t}{\sigma_h}) \frac{\partial k}{\partial x_j} \right] + \frac{G_k}{\rho} - \varepsilon \tag{7}$$

$$\frac{\partial \varepsilon}{\partial t} + u_j \frac{\partial \varepsilon}{\partial x_j} = \frac{1}{\rho} \frac{\partial}{\partial x_j} \left[(\mu + \frac{\mu_t}{\sigma_h}) \frac{\partial \varepsilon}{\partial x_j} \right] + \frac{1}{\rho} C_{\varepsilon 1} G_k \frac{\varepsilon}{k} - C_{\varepsilon 2} \frac{\varepsilon^2}{k} \tag{8}$$

where C_μ, $C_{\varepsilon 1}$, $C_{\varepsilon 2}$, σ_k, and σ_ε are the default k-ε model coefficients. $G_k = \mu_t S_i$ is the generation of turbulent kinetic energy due to mean velocity gradients with S_i the mean rate of strain tensor. The eddy viscosity is given by $\mu_t = C_\mu \rho k^2/\varepsilon$.

2. Numerical Model and Verification

In this study, the multiphase flow model that includes air and tracer gas is selected. By using the mixture model technique of the Fluent, the numerical simulation can predict the tracer gas dispersion in volume fraction for urban street canyon area. In order to obtain the optimum parameters for pollutant dispersion model in urban street canyon, which include the selections of turbulence model, boundary condition, and grid size etc., the different combination of parameter cases are tested. The numerical model used in this study excerpted from the previous researches by the authors (Chang [3], Chang *et al*. [4], and Chang and Meroney [5, 6]).

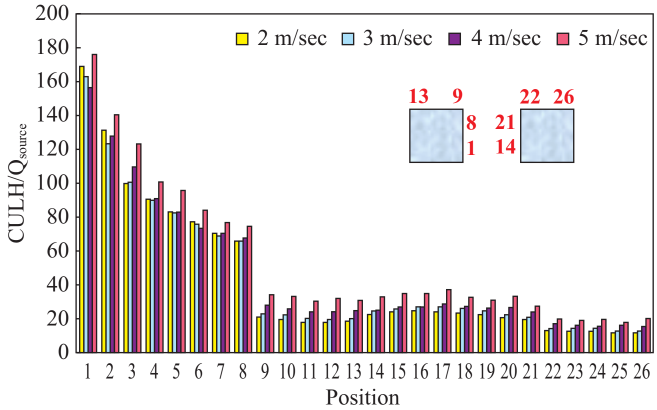

Fig. 4. The effect of different wind velocity on pollutant concentrations in a street canyon. Qs: total pollutant emission (Qsource = Qair (135 l/h) + Qpropane (4 l/h)) (H: canyon height = 8 cm, L: canyon length = 75 cm, C: concentration, U: reference velocity).

IV. RESULT AND DISCUSSION

1. Wind Tunnel Test in a Symmetric Street Canyon and the Analysis of Numerical Simulation

1) The Results from the Wind Tunnel Test

When the street width to building height ratio was 1 (B/H = 1) and the wind speed was changed from 0.5 to 5 m/sec, the results showed that the changes of pollutant concentration were not significant. However, under the same linear pollution source, Meroney reveals that the concentration at the leeward side of the street canyon is twice as high if the aspect ratios were B/H = 2 and B/H = 4. From smoke flow visualization, it can be clearly observed that a more stabilized vortex is formed in the street canyon, but with poor ventilation [16].

Fig. 4 depicts the upwind site (upwind wall area) of the street canyon, from the bottom to the rooftop of the buildings' leeward side (from 1 to 8) and the downwind site (downwind wall area) of the street canyon, from the bottom to the rooftop of the buildings' windward side (from 14 to 21). The dimensionless concentration in the leeward side is 3 or 4 times higher than it is in the windward side. The highest concentration in the leeward side can be found at the location marked as number 1 (around 1 cm above the ground). Its dimensionless concentration tends to decrease as the height of the leeward side increases. However, its dimensionless concentration is still higher than the dimensionless concentration in the windward site.

From left to right, the rooftops of the buildings in the upwind area (from 13 to 9), from left to right, the rooftops of the buildings in the downwind side (number 22 to 26), it can be

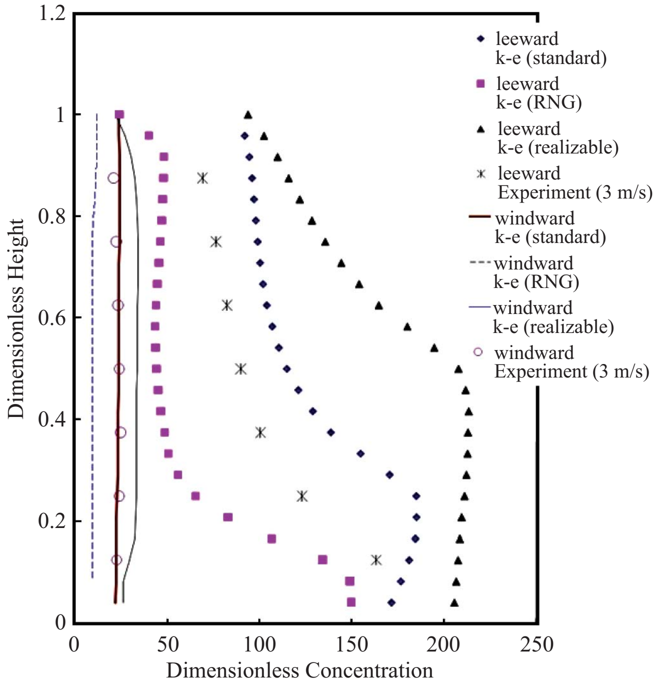

Fig. 5. Comparisons of the wind tunnel test and numerical model concentrations with two dimensional street canyon (B/H = 1).

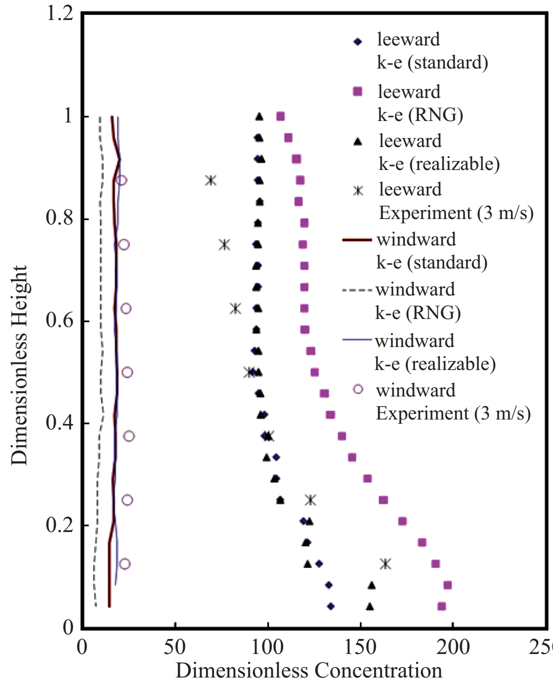

Fig. 6. Comparisons of the wind tunnel test and numerical model concentrations with three dimensional street canyon (B/H = 1).

seen that the dimensionless concentration of the rooftops of the buildings in the upwind site is higher than the dimensionless concentration of the rooftops in the downwind site. The highest dimensionless concentration was detected at the location of number 9. This is because the leeward side happens to have a higher dimensionless concentration than in the windward side. Moreover, the location of number 9 is also close to the leeward side. These were factors that caused a higher dimensionless concentration in the leeward side. Some other noticeable findings were also found in the study. For example, around number 13 (the rooftop in the upwind area), the dimensionless concentration was higher than the locations around number 10 to number 12. The following description explains such a phenomenon. When the roughness surface of the buildings in the upwind area generates separation, It causes vortex on the surfaces of the buildings in the upwind area. Thus, the pollution concentration increases in that area.

2) The Analysis of the Numerical Simulation

Fig. 5 represents that when wind speed was 3 m/sec, the data collected from the buildings' windward side by using standard turbulence model was relatively the same as the data from the physical experiment in the downwind site. When compared, the dimensionless concentration detected from the buildings' leeward side in the upwind site of the street canyon, the experimental results from the standard turbulence model are better than those from the renormalization groups turbulence model. However, the results from the realizable turbulence model have a greater deviation than the results collected from the physical experiment.

Fig. 6 shows that when the wind speed in the three-dimensional street canyon was 3 m/sec, the data collected in the leeward side indicates that the statistics from the standard turbulence model match those from the realizable turbulence model. As for the data from the renormalization groups turbulence model, due to increase in wind speed, the dimensionless concentration also increased. Thus, the results from the renormalizations group turbulence model in the three-dimensional simulation can't be as accurate as it is in the two-dimensional simulation.

The deviations in two-dimensional and three-dimensional leeward simulations (in standard turbulence model) are 5% and 8%. The deviations in the two-dimensional and three-dimensional windward simulations (in standard turbulence model) are 23% and 15%. The results show that when a two-dimensional numerical simulation in the street canyon wind tunnel test is used, a certain degree of accuracy can still be reached.

2. A Street Canyon with the Height of 4, 8, 12, 16 cm

The leeward of the buildings arranged in the downwind site of the street canyon from low to high are rake 1, rake 3, and rake 5, respectively. The windward of the buildings arranged accordingly in the windward site of the street canyon are rake 2, rake 4, and rake 6. Fig. 7 represents that the wind tunnel experimental data and the concentration in the leeward side is higher than the concentration in the windward side. As for rake 1, the variation of the dimensionless concentration is relatively unstable. The increase of the wind speed will lower the concentration due to the fact that the wind can sweep pollutants away.

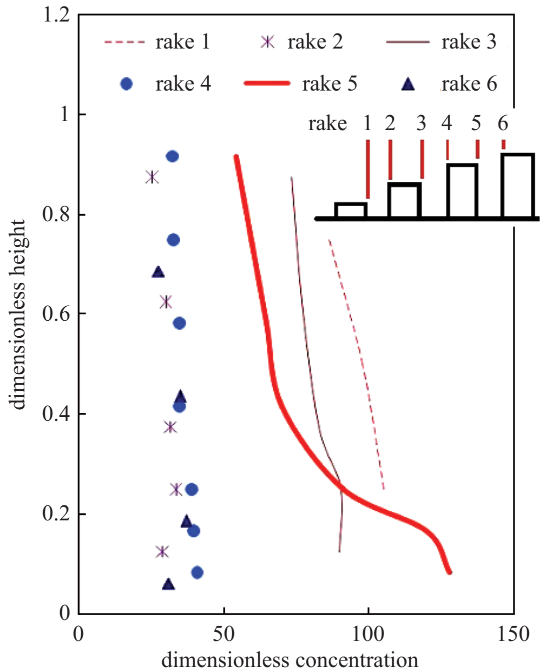

Fig. 7. Concentrations in the canyon for different canyon height (reference velocity = 4 m/sec).

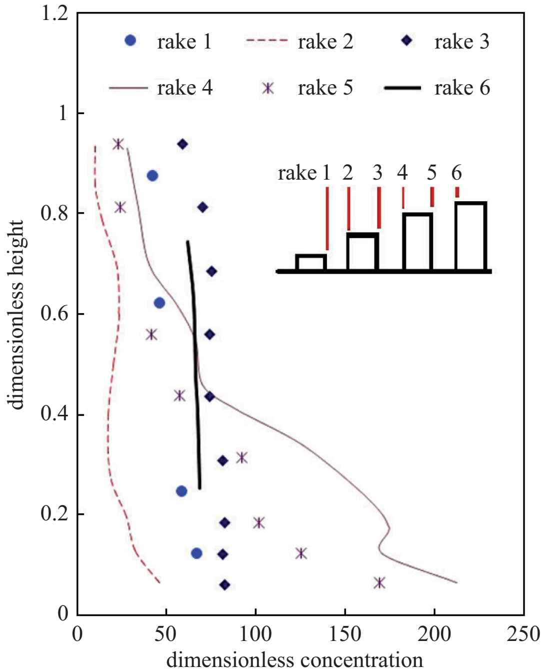

Fig. 9. Concentrations in the canyon for different canyon height (reference velocity = 4 m/sec).

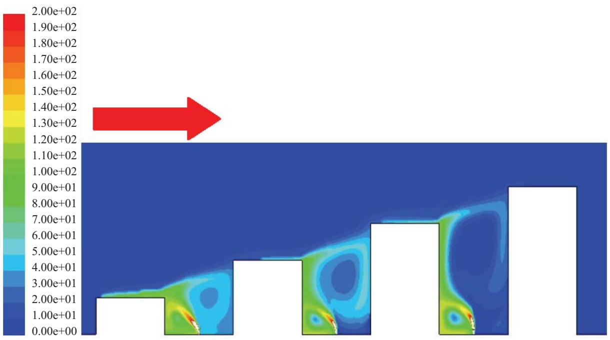

Fig. 8. Numerical model (FLUENT RNG κ-ε) for dimensionless concentration contour (reference velocity = 4 m/sec).

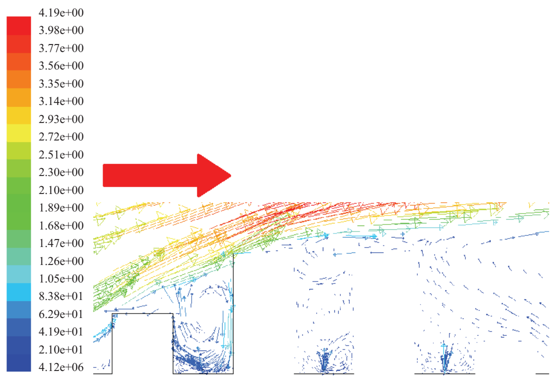

Fig. 10. Numerical model (FLUENT RNG κ-ε) for velocity vector field.

The numerical simulation can also predict dimensionless concentration as Fig. 8 shows. The result depicts that the dimensionless concentration mainly is detected in the leeward side. In other words, the dimensionless concentration in the leeward side is higher than the dimensionless concentration in the windward side. The result also matches the result from the wind tunnel test. The factor that affects the accumulation of pollutants is viscosity.

3. A Street Canyon with the Height of 8, 16, 16, 4 cm

In Figs. 10 and 11, if an 8 cm street canyon was placed in front of a 16 cm symmetric street canyon, the velocity of field flow would be different from a symmetric street canyon. When the flow passes the 8 cm street canyon, the rooftop corner will generate separation flow. When the flow passes the 16 cm rooftop corner, the velocity of separation flow is noticeable. This could also be the reason the direction of the flow is opposite to the direction of the original velocity field in the center of the 16 cm street canyon. Furthermore, in the analysis of the velocity field, the top of rake 3 will generate an anti-clockwise vortex. In the bottom of rake 4, a smaller but stabilized clockwise vortex is generated. Therefore, the bottom of rake 4 has more accumulated pollutants.

From both Fig. 9, the dimensionless concentration can be observed around 0.45 dimensionless heights. The variation of the dimensionless concentration is mainly affected by the anti-clockwise vortex on the top of the street canyon and the clockwise vortex at the bottom of the street canyon. Obviously, the variation of the dimensionless concentration in both rake 3 and rake 4 is different from the data of the symmetric height street canyon (8 cm).

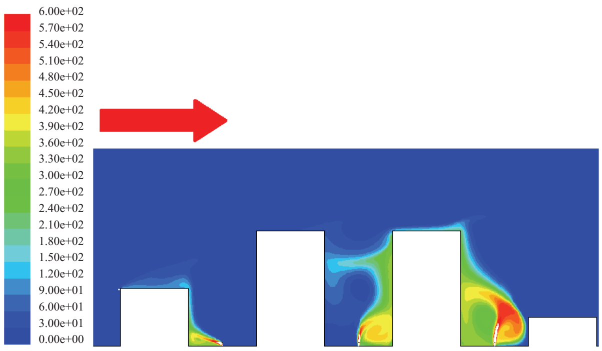

Fig. 11. Numerical model (FLUENT RNG κ-ε) for dimensionless concentration contour.

4. Summary of the Result and Discussion

(1) From the wind tunnel test in the two-dimensional symmetric height street canyon, with various wind speeds, the results showed that the pattern of dimensionless concentration distribution matched the pattern investigated by Meroney [4].

(2) From the prediction of two-dimensional numerical simulation, the data derived from the renormalization groups and the standard turbulence model were better than the results with the realizable turbulence model. This also confirms the results from Chan's study [20]. That is, the results from the renormalization groups turbulence model in the street canyon are more accurate. However, the statistics and the comparison of the concentration on the rooftops were not illustrated by Chan.

(3) Using numerical simulation with a three dimensional symmetric street canyon, the prediction from the standard turbulence model is similar to the results from the physical experiment.

(4) The measurement of the numerical simulation in the renormalization groups turbulence model and the wind tunnel test is precise. Numerical simulation can rapidly simulate the changes of the flow field and the distribution of pollution concentration in a street canyon. However, if the credibility of the simulation is doubtful, a physical experiment using a wind tunnel should be conducted.

(5) In a symmetric height 16 cm street canyon with various heights of 8, 16, 16, 4 cm accordingly, the pattern of the dimensionless concentration did not match the prediction from the symmetric height 16 cm street canyon. In the street canyon with different heights, pollutant transportation and accumulation was considerable. The density of the dimensionless concentration in the nearby areas reached the highest level. The health of people such as pedestrians, residents or workers who live or work in the nearby areas would be seriously affected by the pollutants.

V. CONCLUSION

In this study, a wind tunnel test in a symmetric height two-dimensional street canyon was conducted. The results from the test were compared with the experimental data from a numerical simulation using the software FLUENT 6.3. The experimental data from the numerical simulation was also compared with the results from the wind tunnel test in an asymmetric height street canyon with 4 different heights. After analyzing the collected data, the conclusions are that the results showed that a higher concentration of pollutants accumulates under the leeward of the street canyon due to the occurrence of a clockwise vortex inside the street canyon when the street canyon aspect ratio (B/H) is 2. On the contrary, over the windward of the street canyon, a lower concentration of pollutants accumulates due to the occurrence of an anti-clockwise vortex. The cfd programs reproduced the overall flow fields observed during the measurement program, but it is evident that steady state calculations are not reproducing the intermittent nature of the penetration of elevated flows down into the canyons. This results in situations where the FLUENT cfd concentrations overpredict magnitudes along canyon walls. However, it is found that wall magnitudes can be very sensitive to the rather crude wall boundary conditions incorporated in the program.

REFERENCES

1. Carpenter, P., "Wind speeds in city streets - full scale measurements and comparisons with tunnel test, Recent advances in Wind Engineering," *Proceedings of the 2nd Asia-Pacific Symposium on Wind Engineering, Oxford*, UK, pp. 845-852 (1989).
2. Chan, T. L., Dong, G., Leung, C. W., Cheung, C. S., and Hung, W. T., "Validation of a two-dimensional pollutant dispersion model in an isolation street canyon," *Atmospheric Environment*, Vol. 36, pp. 861-872 (2002).
3. Chang, C. H., "Computational fluid dynamics simulation of concentration distributions from a point source in the urban street canyons," *Journal of Aerospace Engineering, ASCE*, Vol. 19, No. 2, pp. 80-86 (2006).
4. Chang, C. H., Chan, C. C., Cheng, K. J., and Lin., J. S., "Computational fluid dynamics simulation of air exhaust dispersion from negative isolation wards of hospitals," *Engineering Applications of Computational Fluid Mechanics*, Vol. 5, No. 2, pp. 276-285 (2011).
5. Chang, C. H. and Meroney, R. N., "Numberical and physical modeling of bluff body flow and dispersion in urban street canyons," *Journal of Wind Engineering and Industrial Aerodynamics*, Vol. 89, pp. 1325-1334 (2001).
6. Chang, C. H. and Meroney, R. N., "Concentration and flow distributions in urban street canyon: wind tunnel and computational data," *Journal of Wind Engineering and Industrial Aerodynamics*, Vol. 91, pp. 1141-1154 (2003).
7. De Paul, F. T. and Shieh, C. M., "A study of pollutant dispersion in an urban street canyon," Report No. ANL/ER-84-1, Argonne National Laboratory, Argonne, IL (1984).
8. He, J. and Song, C. C. S., "A numerical study of wind flow around the TTU building and the roof corner vortex," *Journal of Wind Engineering and Industrial Aerodynamics*, Vols. 67-68, pp. 547-558 (1997).
9. Hwang, R. R., Chow, Y. C., and Peng, T. F., "numerical study of turbulence flow over two-dimensional surface mounted ribs in a channel," *Journal of Numerical Methods Fluids*, Vol. 31, pp. 767-785 (1999).
10. Johnson, W. B., Dabetdt, W. F., Ludwig, F. L., and Allen, R. J., "Field study for initial evaluation of an urban diffusion model for carbon monoxide," *Comprehensive Report, Coordinating Research Council Contract CAPA-3-68*, Stanford Research Institute, Menlo Park, CA, pp. 1-69 (1971).
11. Johnson, W. B., Ludwig, F. L., Dabberdt, W. F., and Allen, R. J., "An urban diffusion simulation model for carbon monoxide," *Journal of Air*

Pollution Control Association, Vol. 23, pp. 490-498 (1973).
12. Kastner-Klein, P., Fedorovich, E., Ketzel, M., Berkowicz, R., and Britter, R., "The modelingof turbulence from traffic in urban dispersion models - Part II: evaluation against laboratory and full-scale concentration measurements in street canyons," *Environmental Fluid Mechanics*, Vol. 3, pp. 145-172 (2003).
13. Kastner-Klein, P. and Plate, E. J., "Wind-tunnel study of concentration fields in street canyons," *Atmospheric Environment*, Vol. 33, pp. 3937-3979 (1999).
14. Kukkonen, J., Valkonen, E., Walden, J., Koskentalo, T., Aarnio, P., Karppinen, A., Berkowicz, R., and Kartastenpaa, R., "A measurement campaign in a street canyon in Helsinki and comparison of results with predictions of the OSPM model," *Atmospheric Environment*, Vol. 35, pp. 231-243 (2001).
15. Lee, I. Y. and Park, H. M., "Parameterization of the pollutant transport and dispersion in urban street canyons," *Atmospheric Environment*, Vol. 28, pp. 2343-2349 (1994).
16. Lee, S., "Unsteady aerodynamic force prediction on a square cylinder using κ-ε turbulence models," *Journal of Wind Engineering and Industrial Aerodynamics*, Vols. 67-68, pp. 79-90 (1997).
17. Leitl, B. M. and Meroney, R. N., "Car exhaust dispersion in a street canyon. Numerical critique of a wind tunnel experiment," *Journal of Wind Engineering and Industrial Aerodynamics*, Vols. 67-68, pp. 293-304 (1997).
18. Maruyama, T. and Maruyama, Y., "Large eddy simulations around a rectangular prism using artificially generated turbulent flows," *Abstracts of Papers at 3rd International Symposium on Computational Wind Engineering*, University of Birmingham, UK, pp. 11-14 (2000).
19. Meroney, R. N., Pavageau, M., Rafailidis, S., and Schatzman, M., "Study of line source characteristics for 2-D physical modeling of pollutant dispersion in street canyons," *Journal of Wind Engineering and Industrial Aerodynamics*, Vol. 62, pp. 37-56 (1996).
20. Munchow, T., "Untersuchung von Ausbreitungsvorgangen in einer turbulenten Windkanalgren-zschicht - Immissionsbelastung in straben-schluchten," Diplommarbeit report, Hamburg University (1991).
21. Murakami, S., "Comparison of various turbulence models applied to a bluff body," *Journal of Wind Engineering and Industrial Aerodynamics*, Vols. 46-47, pp. 21-36 (1993).
22. Tsai, M. Y. and Chen, K. S., "Measurements and three-dimensional diffusion modeling of air pollutant dispersion in an urban street canyon," *Atmospheric Environment*, Vol. 38, pp. 5911-5924 (2004).
23. Tsuchiya, M., Murakami, S., Mochida, A., Kondo, K., and Ishida, Y., "Development of a new κ-ε model for flow and pressure fields around bluff body," *Journal of Wind Engineering and Industrial Aerodynamics*, Vols. 67-68, pp. 169-182 (1997).
24. Yamartino, R. J. and Wiegand, G., "Development and evaluation of simple models for flow, turbulence and pollutant concentration fields within an urban street canyon," *Atmospheric Environment*, Vol. 20, pp. 2137-2156 (1986).
25. Zhang, Y. Q., Hubert, A. H., Arya, S. P. P., and Snyder, W. H., "Numerical simulation to determine the effects of incident wind shear and turbulence level on the flow around a building," *Journal of Wind Engineering and Industrial Aerodynamics*, Vols. 46-47, pp. 129-134 (1993).