Study on Air Change Rate of Urban Street Model

* Young-Duk Kim(1), Yee-Seog Kang(2), Yohanes Joni T.C(3)

(1) *Environmental Engineering, Kwandong University, Korea, kimyd@kd.ac.kr*
(2) *Industry Academy Cooperation Foundation of Kwandong University, Korea*
(3) *Future Lighting Co.,Ltd, Gangneung City, Korea,*

ABSTRACT

The demand on the automobiles has been rapidly growing owing to the high economic growth and the increase of the level of living standard. Thousands of new elevated structures, other tall buildings and many skyscrapers have been built in many big cities. Those social and life-style changes have created many adverse effects to the environment such as toxic materials, air pollution and other related problems. The purpose of this study is to investigate the air change rate in urban street areas around high-rise compact buildings. As one of methods to test the characteristics of the diffusion of the air pollution through the streets in big cities, the distribution of pollutant concentration is measured on two models and the ventilation rate. In order to understand the CO^2 concentration, the flow and dispersion of gases was simulated using both Wind tunnel and Computational Fluid Dynamics (CFD). According to the result, the width of the street, distance between the buildings and height of buildings give considerable effect to the diffusion of the polluted air within the street.

1. INTRODUCTION

The demand on the automobiles has been rapidly growing owing to the high economic growth and the increase of the level of living standard. Thousands of new elevated structures, other tall buildings and many skyscrapers have been built in many big cities. Those social and life-style changes have created many adverse effects to the environment such as toxic materials, air pollution and other related problems. However, although some relevant laboratories and institutions have pointed out problems related to the diffusion of pollution created by the height and interval of the buildings on both sides of the street and the elevated structures constructed on the center of the street, none of them have thoroughly investigated such problems based on practical researches. High-rise building structures such as the city are very complicated and very many vehicles, so large amounts of air pollutants by vehicles will occur. Diffusion of air pollutants generated to identify the surrounding landscape is very complex because of the complex topography. The easiest way to obtain horizontal or concentration as

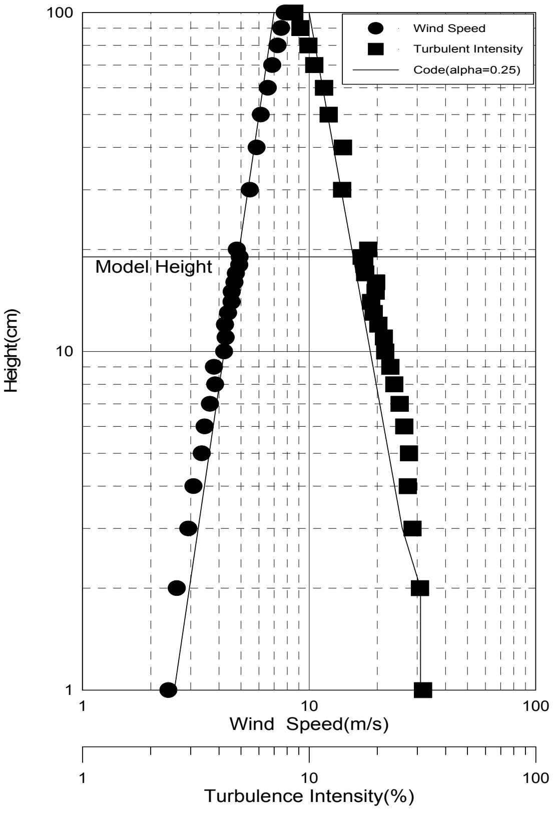
---

(1), (2) Professor
(3) Researcher

mentioned in the introduction is a model of the virtual space. Ventilation when using the model number of the virtual space is the most important. However, the number of air-space model for the virtual case is not much studied experimentally. The purpose of this study is to investigate the air change rate in urban street areas around high-rise compact buildings and the wind based on the wind tunnel test. As one of methods to test the characteristics of the diffusion of the air pollution through the streets in big cities, the distribution of pollutant concentration is measured on two models and the ventilation rate. In addition, by simplifying the urban landscape and surrounding buildings as a model to create various forms of virtual Enclosure ventilation frequency and concentration were determined. 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 are compared against fluid modeling in a small Wind Tunnel at Hyundai Institute of Construction Technology.

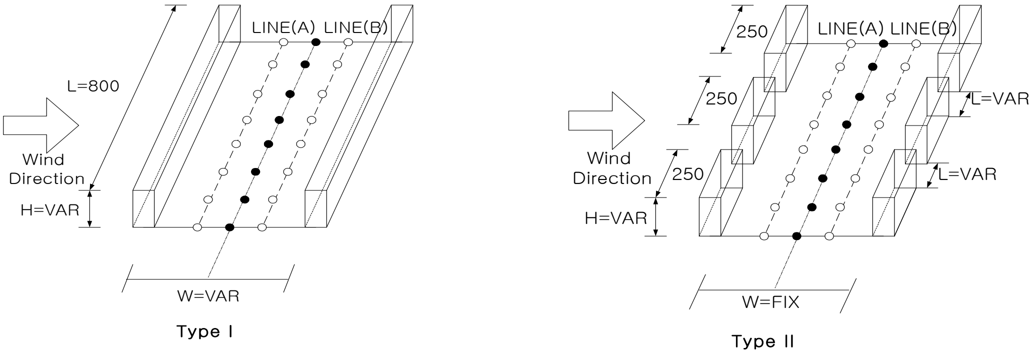
2. EXPERIMENT SETUP AND NUMERICAL SIMULATION

2.1. Wind tunnel test



The wind tunnel used in this study is a small wind tunnel at Hyundai Institute of Construction Technology, which has wind speed of 0.32 to 22.5m/s and the test section of 1.5m height, 1.0m wide and 6.0m long. Wind speed is controlled by adjustment the RPM of the fan and digital manometer and the dynamic pressure was measured using a Pitot-tube. The flow field approaching under condition of metropolitan city is $u \propto z^{1/4}$, turbulence intensity at the same height of model is 18%. In the experiments the model was assumed in the horizontal landscape of the city buildings on both sides that the two types of assumptions were made. In the case of building type I have 800mm length, 50mm width on both sides of the building with variation distances and type II have 250mm long and 50mm width of the building with three units of the building were placed on both sides of the horizontal and 120mm on the gap between two building (fig. 1).

Experiments depending on the type of building height, the gap between buildings and roads were to be adjusted. Measurement model Line (A) and Line (B) is 3mm diameter 4 screws on each pipe was installed, the building mid-height level measuring points in each pipe was installed with 3mm. Gas in the middle of nine 3mm between each building were installed. Overview of the model shown in fig.2



● Gas influent point ○ Gas measuring point

Fig. 1 Scheme of test

Table 1 show the two types of wind tunnel test study, in the type I the width of buildings gap is varies into some several distance and also the different height of buildings. For type II, the buildings gap is the same but different in the interval and height of buildings.

Table 1 types of test

	W(mm)	L(mm)	H(mm)				
			50	75	100	125	150
Type Ⅰ	40		○	○	○	○	○
	120		○	○	○	○	○
Type Ⅱ	120	12.5	○	○	○	○	○
	120	25.0	○	○	○	○	○

Where L : interval of buildings, H : building heights, W : road widths
○ : measuring point - H/2

Concentration was measured by the Trace Gas. CO^2 was used to include trace gas, constantly injecting at 1cc/sec. The concentration of trace gas analyzer (Type 1302 (B & K)) were used. Fig. 2 show the system setup for tracer gas distribution

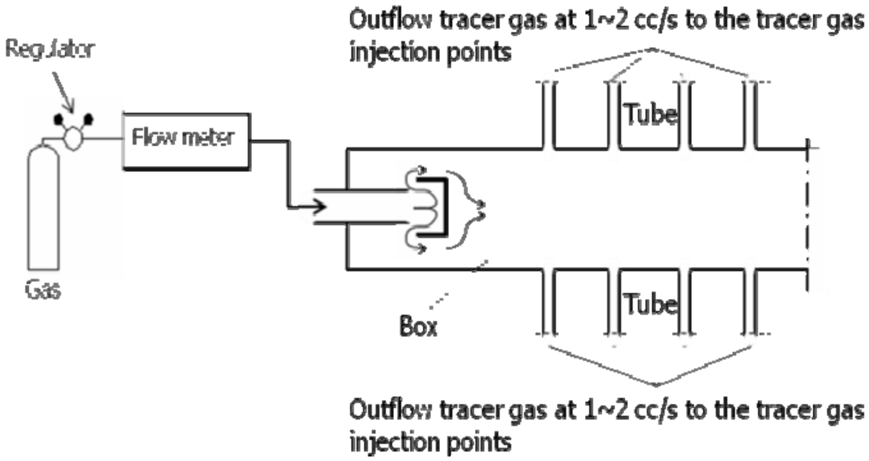


Fig. 2 System setup for tracer gas distribution

2.2. Numerical simulation

ANSYS-CFX was used for numerical simulations. The power law relationship has been used to estimate the wind speed value at the stack level: U = Uo (Z / Zo)^n where, Uo is a reference wind speed (m/s) measured at a certain reference height Zo (m), and n is an index. The value of n depends on the atmospheric stability category and the terrain type (rural or urban (1/4)). A tracer mass fraction of 1 was applied to the source inlet during the calculation. In presenting the results from the calculation, tracer gases concentration was normalized to facilitate comparison with experiments and other numerical results.

Table 2 Parameters and conditions of the calculations

Turbulent model	Standard k-ε model
Difference scheme	Second order upwind
Inlet	$U = U_0 (Z / Z_0)^{1/4}$ $K = 1.5(I \times U)^2, I = 0.1$ $\varepsilon = C_\mu k^{3/2} / l$ $l = 4(C_\mu K)^{1/2} Z_0 Z^{3/4} / U_0$
Pollution Source strengths	Road: CO^2 = 1cc/s
Side, sky	Free slip
Wall	Generalized logarithmic law

Since the area of study contains simple geometric configurations, it was preferred to use a system of unstructured grid because this type of meshes is very effective for conducting CFD simulations in complex urban areas. Thus, an unstructured grid system including 1,100,000 meshes was created. The generalized logarithmic law was applied to the building walls and ground surface as smooth walls, while the side and sky boundaries were treated as free slip surfaces. With respect to inlet boundary, the constant flux layer assumption was adopted to generate a turbulent energy ‘k’ and dissipation rate ‘ε’. The inlet wind speed was set to obey the one-fourth power law relationship and the inlet turbulent intensity was assumed to be 10 % of the reference wind speed. A user subroutine was implemented to specify all of the above quantities at the inlet boundary. The Navier-Stokes equations were solved using the CFD code, ANSYS-CFX, where the standard k-ε model for turbulence was applied. Steady-state analysis was adopted and the second upwind difference scheme was applied to the spatial difference since that scheme is specially adapted for un-structured meshes

2.3. Point sources

The concentration of the gaseous pollutants due to a certain point source is given by the following equation

Non dimensional Tracer gas concentration ($C_{i(n)}$)

$$C_{i(n)} = \frac{C_i}{C_0}\left[\frac{ppm}{ppm}\right] \quad \text{..}(1)$$

Where $C_i =$ Measured tracer gas concentration at each point [ppm]

$$C_0[ppm] = \frac{q}{U_{ref}.H^2} \quad \text{...}(2)$$

Where: $q =$ Tracer gas quantity [cc/s] $U_{ref} =$ Velocity at building height [m/s], $H =$ Building height [m]

Air change rates (N)

$$\text{..} \quad N = \frac{Q}{V} \quad \text{...}(3)$$

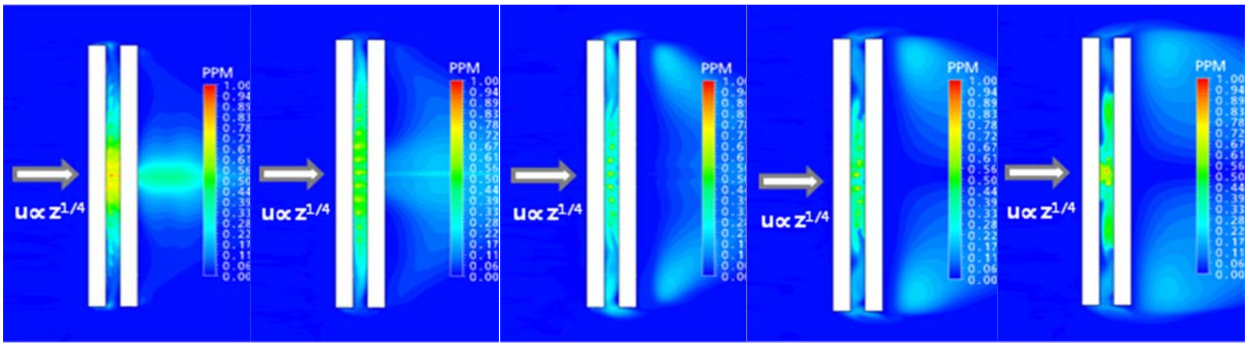
Where Q (Ventilation Rate) = q(cc/sec)/Ca(ppm), q = Total tracer gas quantity (cc/sec), Ca = Average measured concentrations (h/2), V = Volume in the horizontal space (m^3)

3. RESULT AND DISCUSSION

3.1 Concentration

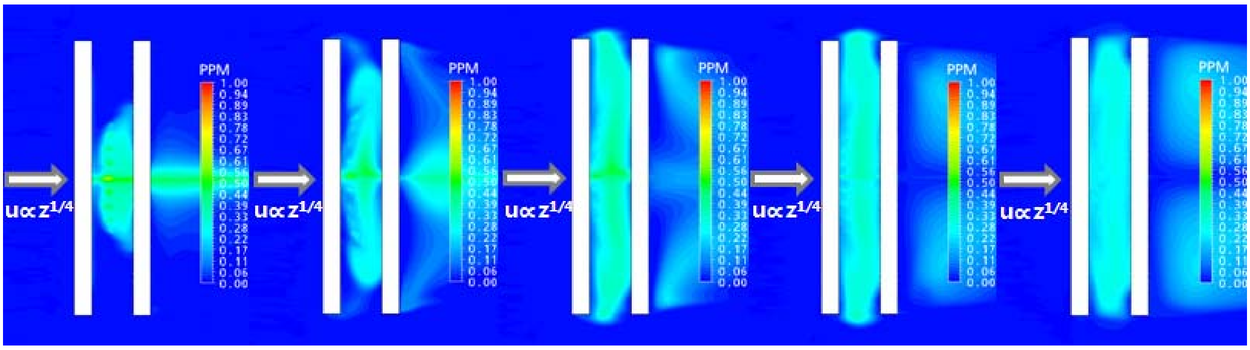
3.1.1 Type I concentration.

Distribution characteristics of CO^2 of type I are presented in fig. 3 and 4 in which the concentration values obtained using CFD are displayed. The figures show high concentration values for CO^2 between two complex buildings. Also it is clear that the concentration larger follow by higher of complex building. As the distance away from the road increases in fig. 4 with 120mm distance, the concentration decrease gradually than fig. 3 with 40mm distance, which means that the pollutants are first emitted from the road and then transport away with wind flow and disperse in the atmospheric air (fig. 4).



a. H = 50 mm b. H = 75 mm c. H = 100 mm d. H = 125 mm e. H = 150 mm

Fig. 3 CO² Concentration fields in location between 40mm width at H/2 plan



a. H = 50 mm b. H = 75 mm c. H = 100 mm d. H = 125 mm e. H = 150 mm

Fig. 4 CO² Concentration fields is in location between 120mm width at H/2 plane

To compare the performance of the experiment against the CFD approach, the comparison of concentration of CO² calculated by the two approaches at measuring points at H/2. Fig. 5 shows concentration of CO² according to building heights (Type I). It is show that the concentration values at 40mm width (1.8 – 13) of the the road between two complex buildings are higher concentration (C / Co) than width of 120 mm (0.3 – 5.9) in the Experiment. Also it is show that the trend results from CFD approach are similar to the experiment.

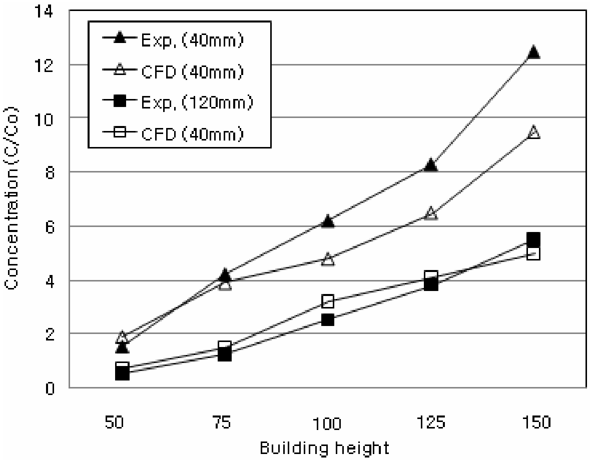
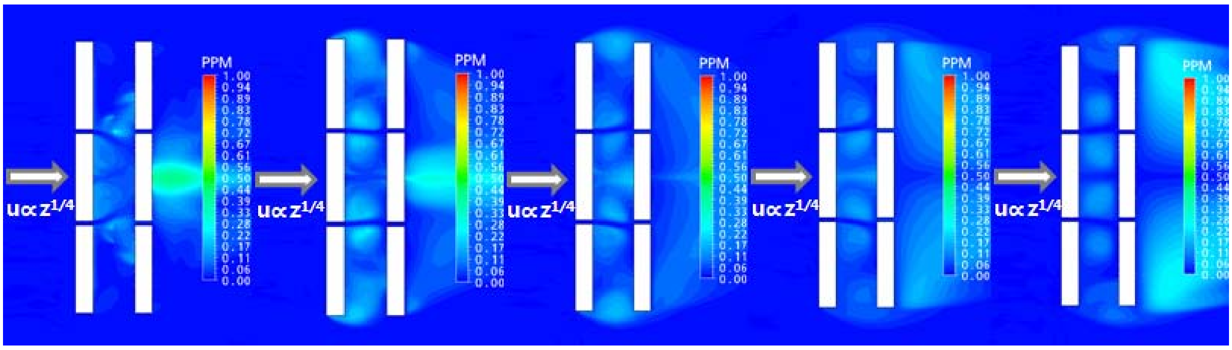


Fig. 5 Concentration of CO2 according to building heights (Type I)

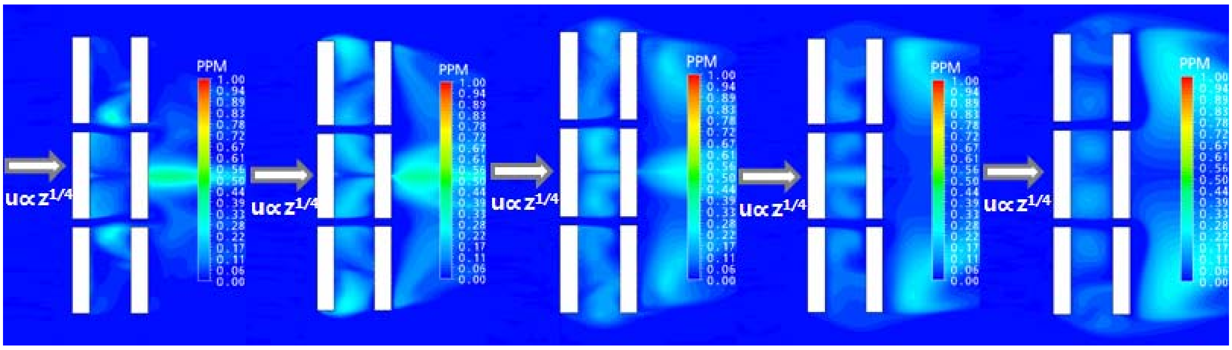
3.1.2 Type II concentration distribution in the (W = 120mm)

Fig. 6 and 7 shows distribution characteristics of CO^2 of type II which the concentration values obtained using CFD are displayed. The figures show higher building is also high concentration values for CO^2 between two complex buildings with three units of the building were placed on both sides of the horizontal. In fig. 7 show that when the distance away from other building, the concentration decrease gradually at distance 25mm compare to the fig. 6 with 12.5 mm, which means that the pollutants are first emitted from the road and then transport away with wind flow and disperse in the atmospheric air.



a. H = 50 mm b. H = 75 mm c. H = 100 mm d. H = 125 mm e. H = 150 mm

Fig. 6 CO^2 Concentration fields in location between 120mm width (L=12.5mm) at H/2 plane



a. H = 50 mm b. H = 75 mm c. H = 100 mm d. H = 125 mm e. H = 150 mm

Fig. 7 CO^2 Concentration fields in location between 120mm width (L=25mm) at H/2 plane

To compare the performance of the experiment against the CFD approach, the comparison of concentration of CO^2 calculated by the two approaches at measuring points at H/2. Fig. 8 shows concentration of CO^2 according to building heights (Type II). It is show that the concentration values at 12.5mm width (0.4 – 3.5) of the road between two complex building has higher concentration (C / Co) than width of 25 mm (0.3 – 2.5) in the Experiment. The trend of CFD approach also show similar results with the experiment.

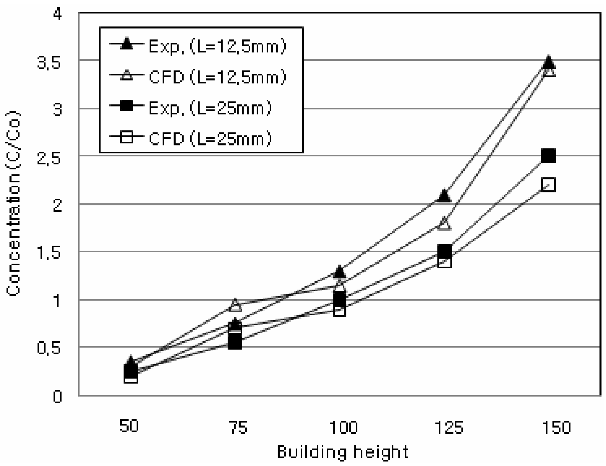


Fig. 8 Concentration of CO2 according to building heights (Type II)

3.2 Air change rate

Table 3 & 4 shows the number of ventilation times at wind speed 1m/s at building heights of type I and II calculated by experiment and CFD approaches. For the overall results of type I range from 1000 – 3400 respectively and type II range from 1600 – 5300 ventilation times it is almost 2 times larger than Type I.

Table 3 Number of ventilation times (type I)
(Wind Speed: 1m/s at building heights)

Type		Ventilation (Exp.)(m^3/hr)	Ventilation (CFD)(m^3/hr)	Ventilation times (Exp.) (time/hr)	Ventilation times (CFD) (time/hr)
Type Ⅰ	H=50mm W=40mm	5.358	5.833	3348.75	3645.89
	H=50mm W=120mm	14.052	16.575	2927.5	3453.22
	H=75mm W=40mm	4.766	5.390	1985.83	2245.54
	H=75mm W=120mm	15.855	18.37	2202.08	2535.56
	H=100mm W=40mm	5.721	5.935	1787.81	1854.89
	H=100mm W=120mm	14.50	16.554	1510.42	1724.33
	H=125mm W=40mm	6.753	6.103	1688.25	1525.65
	H=125mm W=120mm	14.927	16.143	1243.92	1345.21
	H=150mm W=40mm	6.520	7.044	1358.33	1467.25
	H=150mm W=120mm	14.419	16.283	1001.32	1130.75

Table 4 Number of ventilation times of type II
(Wind speed: 1m/s at building heights)

Type		**Ventilation (Exp.)(m3/hr)**	**Ventilation (CFD)(m3/hr)**	**Ventilation times (Exp.) (time/hr)**	**Ventilation times (CFD) (time/hr)**
Type Ⅱ (W=120mm)	H=50mm L=12.5mm	25.476	27.520	5307.5	5732.54
	H=50mm L=25 mm	38.485	40.512	8107.71	8534.77
	H=75mm L=12.5mm	26.259	26.719	3647.08	3710.78
	H=75mm L=25mm	36.179	39.922	5024.86	5544.43
	H=100mm L=12.5mm	26.921	28.859	2804.27	3001.34
	H=100mm L=25mm	35.394	36.618	3686.88	3844.99
	H=125mm L=12.5mm	26.771	27.933	2230.92	2327.69
	H=125mm L=25mm	37.706	42.286	3142.17	3523.76
	H=150mm L=12.5mm	23.073	26.729	1602.29	1856.08
	H=150mm L=25mm	32.188	36.786	2235.28	2554.44

To compare the performance of the experiment against the CFD approach, the number of ventilation times according to building heights of type I and II were compared. Fig. 9 show the number of ventilation times according to building heights calculated by experiment and CFD approaches. As the results, in the case of 40 and 120mm width the values in the range of 1000 – 3500 number of ventilation time and it is show that the higher number of ventilation, the building height was lower.

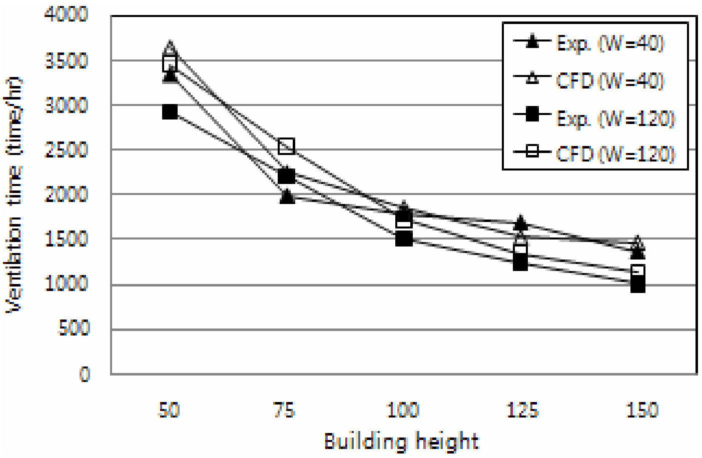


Fig. 9 Number of ventilation times according to building heights

Fig. 10 Fig. 9 show the number of ventilation times according to building heights calculated by experiment and CFD approaches. As the results, in the case of L=12.5 and L=25mm width the values in the range of 1500 – 8100 number of ventilation time and also show that the higher of the building, the number of ventilation was decrease.

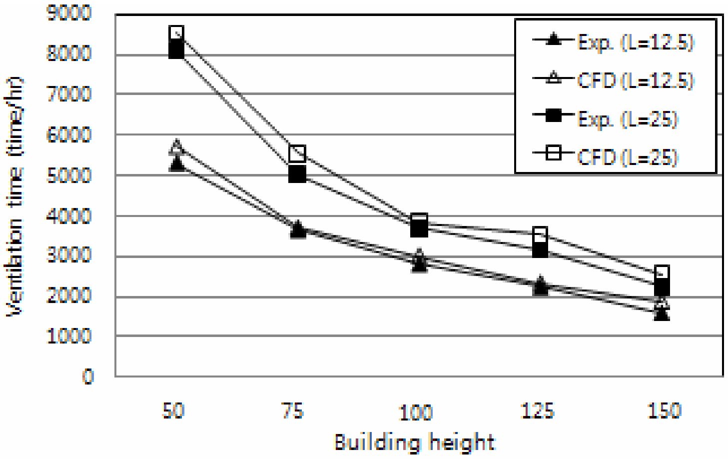


Fig. 10 Number of ventilation times according to building heights

CONCLUSION

In this study, a wind tunnel test in a symmetric height street canyon was conducted. The results from the test were compared with the experimental data from a numerical simulation using the software ANSYS-CFX From the three dimensional symmetric street canyon CFD modeling; the prediction from the standard turbulence model is similar to the results from the wind tunnel. From the tests in a street canyon with various heights from high to low and in a street canyon with various heights from low to high, the results indicated that with different height arrangements the geometric shapes changed directions of the flow fields. Eventually, accumulation of the dimensionless concentration would also change. CFD simulation can simulate the changes of the flow field and the distribution of pollution concentration in a street canyon. In the street canyon with different heights, pollutant transportation and accumulation was considerable. 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 pollution.

Acknowledgment

The work described in this paper was partially supported by RIC(N) of Kwandong University

REFERENCES

Yee-Seog Kang, “ A Study on Characteristics and Practicalities of Atmospheric Diffusion and Environmental Obstacle Induced by Wind”

ASCE, “Wind tunnel model studies of Buildings and Structures", ASCE Manuals and Reports on Engineering Practice No. 67, 1987

Nobuyuki Kobayashi et al., A Wind Tunnel Model Study of the Diffusion of Exhausts around Building in an Urban Area and its Environmental Impact.

H. B. “AWBI and M. M. NEMRI, Scale Effect in Room Airflow studies", Energy and Buildings, 14 pp.207-210, 1990.

H. L. Higson et al., Concentration Measurements around an Isolated Building, a Comparison between Wind Tunnel and Field data", Atmospheric Environment, Vol. 28, No. 11, pp.1827-1836, 1994.

Jin, S. and K. Demerjian, “A Photochemical Box Model for urban air quality study", Atmospheric Environment, 27B, 371-387, 1993

Huber, A. H. Wind tunnel and Gaussian plume modelling of building wake dispersion, Atmospheric Environment, 25A, 1237-1249, 1991.

Verver, G.H.L. and F.A.A.M. De Leeuw An operational puff dispersion model, Atmospheric Environment, 26A, 3179-31193, 1992.

A. G. Davenport Approach to human comfort criteria for environmental wind condition, Colloquium on Building Climatology, Stockholm, 1972.

F. A. Gifford, “Turbulent Diffusion - Typing Schemes, A Review”, Nuclear Safety, Vol. 17, No. 1, pp. 68-86, 1976.

Henry W. Tieleman, Wind Tunnel Simulation of Turbulence in the Surface Layer, J. of Wind Engineering and Industrial Aerodynamics, 36, pp. 1309-1318, 1990.

J. A. Peeter and J. E. Cermark, Simulation of Atmospheric Flows in Short Wind Tunnel Test Section.

Jie Xuan and Alan Robins, “The Effects of Turbulence and Complex on Dust Emissions and Deposition from coal Stockpiles", Atmospheric Environment, Vol. 28, No. 11, pp.1951-1960, 1994.