

Comparative study of concentrations and distributions of CO and NO in an urban area: Gaussian plume model and CFD analysis

M. Bady¹, S. Kato², R. Ooka², H. Huang² & T. Jiang¹

¹*Graduate School of Engineering, The University of Tokyo, Japan*

²*Institute of Industrial Science, The University of Tokyo, Japan*

Abstract

The dispersion of air pollutants, CO and NO, due to three point sources and a busy traffic road in a certain area located in the central part of Tokyo has been studied using two different approaches. The first approach was based on the well known Gaussian plume model (GPM) while the second approach was a numerical modelling using a CFD code. In the first approach, the domain of study was divided into a fine grid and the line source (road) was treated as a number of continuous adjacent stacks. The number of ground level stacks was taken according to the length of the road and the diameter of each stack was set equal to the width of the road. The GPM of a point source was applied upon each stack and the concentration at any receptor was calculated by applying the principle of superposition. In the second approach, numerical analysis was performed using the CFD code STAR-CD in order to simulate the wind flow over the area considered by applying the standard k- ϵ turbulent model. Steady state analysis was adopted and the second order upwind difference scheme was applied to the spatial difference. Wind velocity in the first approach, was set equal to the velocity at 10 m height for the ground level line source, and equal to the velocity value at the stack height level for the three tall stacks, while in the numerical approach, the wind velocity was obeying the power law relationship. Results of the two approaches were compared together and their results have shown the same trend for the distributions of CO and NO concentrations inside the area of study. The agreement between the concentration values calculated by the two approaches was good in less densely built-up locations inside the domain of study, and it was not accepted in the high densely built-up sites.

Keywords: Gaussian plume model, CFD, pollutant concentration, CO and NO.



1 Introduction

During the last few decades, investigation of air pollutants transportation and dispersion in urban areas has become of major concern for the protection of air quality and also for compliance with air pollution regulations. The importance of air pollutants investigation arises from the increased levels of these pollutants in the atmospheric air of urban areas. Such increase has many reasons such as the continuing expansion of industries and the increased usage of motor vehicles coupled with population growth, especially in large urban areas. These sources introduce a variety of air pollutants in large quantities into the atmosphere.

Although, the ambient air quality is monitored regularly to check the pollutants levels by the regulatory agencies, regular monitoring of pollutants is not always feasible and it is cumbersome and costly. Thus, air pollution modeling has become an indispensable tool to assess the ambient air quality and to maintain pollutants level within permissible limits. In addition to that, the ability to predict ground level concentrations of air pollutants is required in order to determine the environmental impact of existing sources to evaluate alternative new source locations, designs and controls and to estimate the effect of possible modifications to existing sources. Also, the impacts of new sources that do not yet exist can only be determined through modeling. Thus, air pollution modeling has become a primary analytical tool in most air quality assessments.

Among many air quality models, Gaussian plume model (GPM) is the most widely used approach for estimating the impact of air pollutants. On the other hand, although numerical modeling requires much more extensive input data and consumes a lot of time, it is considered more appropriate than GPM for urban areas applications.

In the present study, a comparison between the performance of GPM approach and a numerical approach, through an analysis for the air pollution problem in a certain urban area located in the central part of Tokyo city, has been carried out. The study was focused on the concentrations of carbon monoxide (CO) and nitrogen oxide (NO) only, since these two gases were identified as the major pollutants which are being emitted from the above sources.

2 Study area

The considered area of study, as displayed in fig. 1, has many characteristics which gave us an attention to study its air pollution problem. The primary source of air pollution within that area is a high density traffic road. It consists of two parts; the first part extends approximately to the north and it has a length of 365 m, while the second part makes an angle of 58° with the south and its length is 333 m. Also, there are three ventilation towers (through the traffic road) for the underground tunnel under the road. These three towers were treated in the analysis as three tall stacks. Considering a certain origin, the coordinates of the three stacks are: (0.8, 41), (30, -52), (64, -106). Each one of these stacks has a physical height of 45 m and an exit area of 50 m^2 . Transportation through the two sides of the traffic road and the emitted gases from the three tall stacks play a major role in the problem of air pollution inside that region.



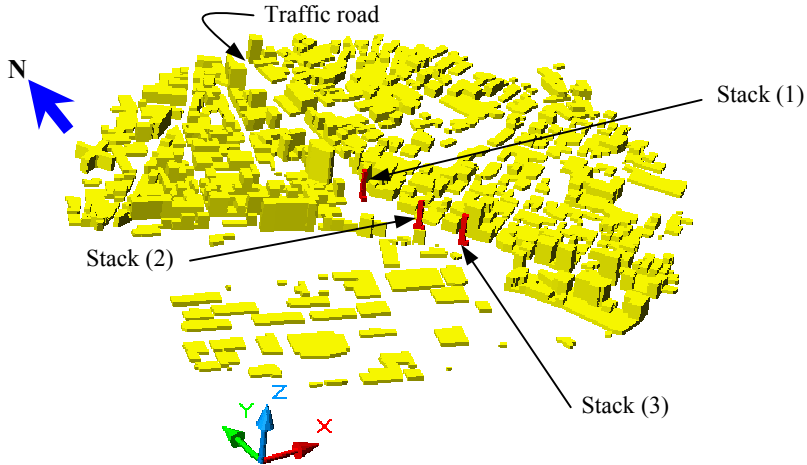


Figure 1: Characteristics of the study area.

3 Methodology

For the purpose of study, a domain of $800 \times 800 \text{ m}^2$ area was considered to calculate the concentrations and distributions of CO and NO inside the domain of study using two different approaches. The first approach was based on the GPM, while the second approach was a numerical simulation using a CFD code. Characteristics of the two approaches are given in the following section.

3.1 Gaussian plume model

To apply Gaussian plume model, the domain of study was simulated using a large number of grids of $5 \text{ m} \times 5 \text{ m}$ size and the traffic road was divided into a current of continuous adjacent stacks. In both cases of the ventilation towers and the traffic road, the origin was transferred to the center of each stack and the traffic road, the origin was transferred to the center of each stack and the modified x-axis was set in the same direction of the wind, while the modified y-axis was taken perpendicular to the wind direction. For any receptor located within the area of study, the downwind and crosswind distances were computed relative to the new origin. Then, the GPM was applied for each point source (3 tall stacks and all ground level point sources which represent the traffic road).

3.1.1 Point sources

The concentration of the gaseous pollutants due to a certain point source is given by the following equation Heinsohn and Kabel [1]:

$$C' = \frac{Q}{2\pi U \sigma_y' \sigma_z'} * \exp\left(-\frac{y'^2}{2\sigma_y'^2}\right) \left\{ \exp\left(-\frac{(Z-H)^2}{2\sigma_z'^2}\right) + \exp\left(-\frac{(Z+H)^2}{2\sigma_z'^2}\right) \right\} \quad (1)$$

where Q denotes the source strength (g/s), U is the mean wind speed (m/s), y' is the crosswind distance (m), Z is the vertical distance above the ground (m), σ_y'

and σ'_z are the horizontal and vertical dispersion parameters (m) respectively and H is the effective stack height (m) which is given as: $H = h + \Delta h$, where h is the physical stack height (m) and Δh is the plume rise (m).

The plume rise can be calculated from Holland's formula Sofuoglu [2]:

$$\Delta h = \frac{V_s D}{U} \left[1.5 + 2.68 * 10^{-3} \cdot p \cdot D \cdot \frac{(T_s - T_a)}{T_s} \right] \quad (2)$$

where V_s is the stack exit gas velocity (m/s), D is the stack exit diameter (m), p is the atmospheric pressure (millibars), T_s is the stack exit gas temperature and T_a is the ambient temperature (K). The prime (') symbol indicates the parameters in wind coordinate system.

The power law relationship has been used to estimate the wind speed value at the stack level:

$$U = U_o \left(Z / Z_o \right)^n \quad (3)$$

where, U_o is a reference wind speed (m/s) measured at a certain reference height Z_o (m), and n is an index. The value of n depends on the atmospheric stability category and the terrain type (rural or urban) Heinsohn and Kabel [1]. Horizontal and vertical dispersion parameters σ_y and σ_z are estimated using the Briggs formulae for urban sites (Griffiths [3]).

3.1.2 Line source (road)

For infinite continuous line sources, eqn.1 can be integrated to obtain an expression for the concentration at any point in the domain around the road Ashoch and Patil [4]. The resulted equation has many restrictions related to wind direction and it needs a correction factor for the value of the used wind speed (Chock [5]). For definite line sources, the problem is different. Usually, it is preferable to treat the traffic road as a line that contains a large number of continuous point sources of the same characteristics of the road, i.e. emission strength. Following that way, the line source in the present study was divided into a current of continuous adjacent point sources as illustrated in fig. 2. The number of point sources representing the traffic road was proportional to the road length, and the diameter of each stack was assumed to be equal to the width of the road. The mechanism of diffusion from each point source was assumed to be independent of the presence of other point sources. The concentration at any receptor was computed by applying the principle of superposition which states that, the concentration at a receptor located at a certain point $P(x,y,z)$, is the sum of contributions from all the point sources making up the line source.

Usually, wind speed value used in concentration calculations is taken as the mean value at the same height of the line source but in the case of ground level roads, it is difficult to measure wind speed at such a small source height. Therefore, wind speed at a height of 10 m above the ground level would be used. It was estimated using eqn. 3.

3.2 CFD modelling

CFD modeling is a general term used to describe the analysis of systems involving fluid flow, heat transfer and associated phenomena by means of



computer based numerical methods. With ever-increasing computer power, computational fluid dynamics techniques have been increasingly applied to the simulation of atmospheric problems Xiaomin *et al.* [6]. In the present study, CFD modeling is based on the numerical solution of the governing fluid flow equations, namely, Navier-Stokes equations and the dispersion equations.

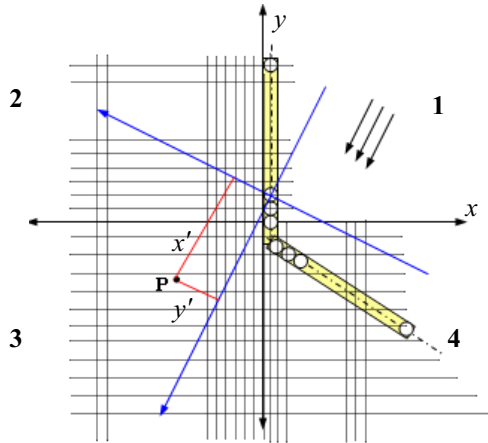


Figure 2: Representation of the line source and the grid.

3.2.1 The mathematical model

The most widely used approach for solving the NS equations is the time averaged, in which these equations are transformed as the Reynolds Averaged Navier-Stokes (RANS) set provided that the widely and practically useful approach, standard k - ϵ turbulent model, was adopted for solutions. Concerning the simulation of CO and NO, they were treated as two active scalars with fixed mass fluxes from the traffic road and the ventilation towers.

3.2.2 Grid and boundary conditions

Since the area of study contains many geometric configurations such as residential houses and shopping centers, 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 Huang *et al.* [7]. Thus, an unstructured grid system including 1,090,712 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.

3.2.3 Computational procedure

The Navier-Stokes equations were solved using the CFD code, STAR-CD, 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 [8]. The SIMPLE algorithm was applied for the calculations of velocity and pressure. The chosen convergence criterion was that the residuals for all calculated variables decreased at each cell below a value of 10^{-6} for the calculations.

4 Simulation data

Table 1 lists all data which has been used in calculating the concentrations of CO and NO inside the study area. Sources strength data and ventilation towers parameters were measured at the actual filed of study. The measured quantities were specifically designed to produce data suitable for CFD simulation and also for GPM calculations. By applying these data and the appropriate parameters, the calculations of CO and NO concentrations were performed using FORTRAN language for the GPM approach and STAR-CD code for the CFD approach. In both approaches, the calculations were performed at a height of 1.5 m above the ground surface (pedestrian level).

Table 1: Parameters and conditions of the calculations.

Inlet	$U = U_o (Z/Z_o)^\alpha, \alpha = 1/4$ $k = 1.5 (I \cdot U)^2, I = 10 \% \text{ \& } \epsilon = C_\mu k^{3/2} / l$ $l = 4 (C_\mu \cdot k)^{1/2} \cdot Z_o \cdot Z^{3/4} / U_o$ Murakami <i>et al.</i> [9] Inlet wind direction: NNE, 67.5° with x-axis.
Pollution source strengths	Road: NO = 1.290 g/s, CO = 0.710 g/s. Tower1: NO = 0.488 g/s, CO = 0.453 g/s. Tower2: NO = 0.385 g/s, CO = 0.359 g/s. Tower3: NO = 0.925 g/s, CO = 0.860 g/s.
Emission parameters of the three ventilation towers	Tip exit velocity = 6.3 m/s. Physical height = 45 m. Exit area = 50 m ² (square stacks). Exit temperature = T_{atm} (assumption).

5 Results and discussion

Figure 3 presents the CFD simulation results of the horizontal wind velocity vector distribution at 1.5 m height above the study area. The average speed in the open zone, where no building exists, is about 1.8 m/s, while it decreases significantly to about 0.3 m/s in the building colony due to the blocking effect of wind flow produced by buildings.



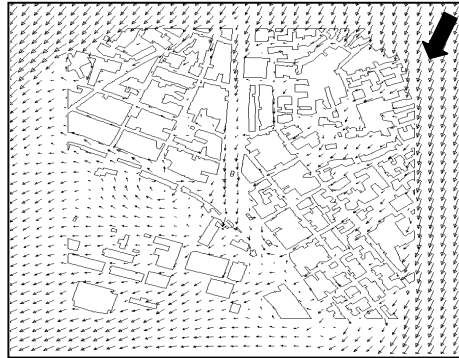
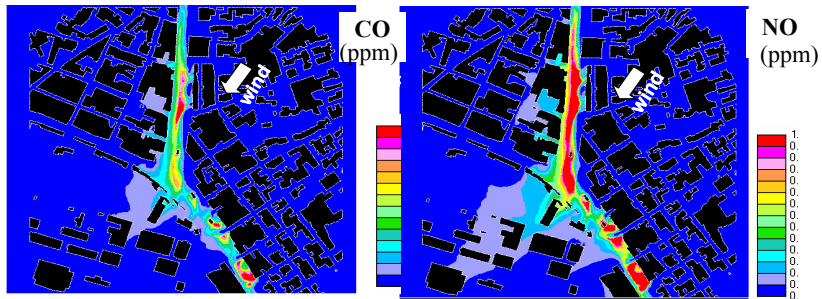
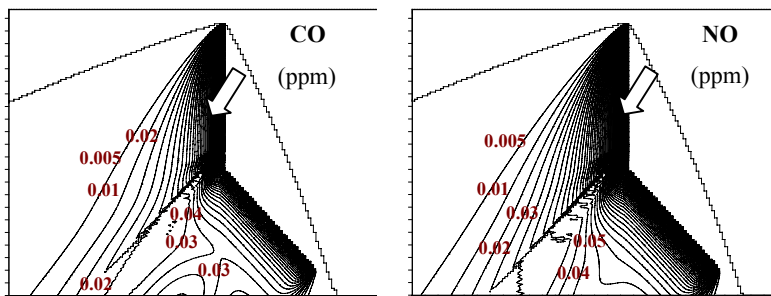


Figure 3: Velocity vector distribution around the area of study.

Distribution characteristics of CO and NO are presented in fig. 4(a) and 4(b) in which the concentration values obtained using CFD are displayed in fig. 4(a) and the concentration contour lines calculated by GPM are shown in fig.4 (b).



(a) CFD



(b) GPM

Figure 4: CO and NO concentrations inside the domain of study.

The above figures show high concentration values for CO and NO_x through the two parts of the traffic road. Also, it is clear that the concentrations in the areas near the traffic road are significantly higher than that of any other area. As

the distance away from the road increases, the concentration decrease gradually which means that the pollutants are first emitted from the road and then transport away with wind flow and disperse in the atmospheric air. Also, it is clear that the average concentrations of NO is reasonably greater than CO concentrations since the emission intensity of NO is greater than that of CO as given in table 1.

5.1 Comparison between the results of the two approaches

To compare the performance of the GPM approach against the CFD approach, the field of study was divided as illustrated in fig. 2 into four quarters. As no concentrations exist in the first quarter, the comparison was conducted for the other three quarters. Figure 5 presents a comparison between the concentrations of CO and NO calculated by the two approaches at some common points located inside the three quarters.

To carry out the comparison between the results of the CFD approach and the GPM approach, it was found that the number of common points in the two approaches is 101 points (34 in the second quarter, 24 in the third quarter and 43 in the fourth quarter). As shown in fig. 5, a significant difference is observed between the concentrations estimated by the two approaches in the case where the building density is high such as the situation in the second quarter. The absolute difference between the concentrations computed by the two approaches is more than 100 % at almost all points inside that quarter. In fact, that difference reflects the disadvantage of using GPM to predict the concentrations in cases of high densely built-up areas since that model doesn't take the buildings effect into consideration. As buildings and obstacles influence both the wind speed and wind direction within the domain around them, the diffusion of air pollutants in that domain is affected (the GPM assumes constant wind direction between the source and the receptor and it doesn't account for the direction variation that can happen due to any obstacle). That can explain the lower concentration values estimated through the GPM compared with that obtained through CFD approach. In the second case where the building density is medium such as the situation of the third quarter, the agreement between the results of the two approaches is acceptable as the deviation lies under an average value of 55 % for CO and 59 % for NO. A very good agreement is observed in the areas where no buildings exist or where the building density is very small such as the case of the fourth quarter. That result show the advantage of using GPM over the numerical modelling (regarding the modelling time and effort) in predicting the concentrations of air pollutants in cases of open sites where no obstacles exist, or where the number of buildings is limited in such away that it makes no resistance to the wind flow.

5.2 Evaluation of the GPM

Beside the disadvantage of the GPM in predicting the actual concentration values in case of high densely built-up areas, the difference between the results of the two approaches in the above cases can be attributed to the following reasons:

(1) Both the intersection region between the two parts of the road and the edge effects caused by the end of the line source was not accounted for in the analysis. These may affect on the concentration calculations since the edge effects become



more important in the sense that they extend to greater crosswind distances as the distance downwind from the source to the receptor increases.

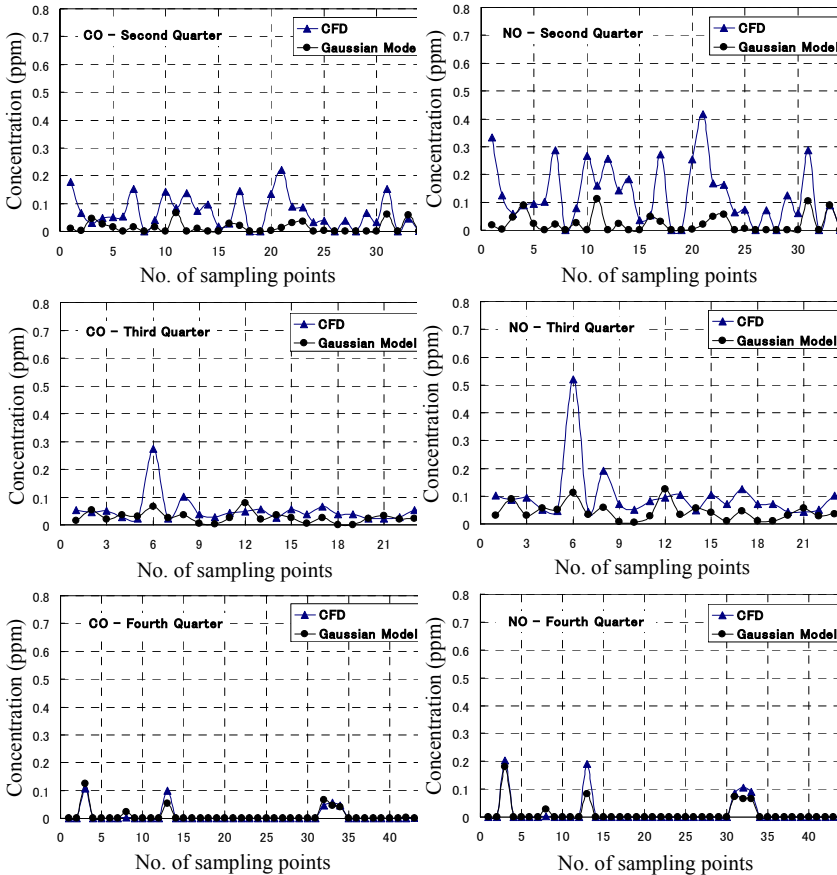


Figure 5: Comparison of GPM and CFD results of CO and NO at different points located within the three quarters of the study area.

(2) As mentioned before, it is difficult to measure the wind speed at the same level of the traffic road. That difficulty drives us to estimate another different value; usually taken as the speed at a height of 10 m. Since the value of $U_z=10$ is greater than the value of $U_z=0$, the pollutant concentration calculated by the GPM at a certain position is lower than the value calculated through the CFD approach at the same position.

(3) The two assumptions of neutral atmospheric conditions and the ventilation towers exit temperatures to be equal to the ambient temperature, decrease the value of the plume rise term. That effect increases the concentration in the near area around the road and decreases it as the distance from the road increases.

(4) Dividing the line source to a current of ground level point sources and using the assumption which states that the mechanism of diffusion from each point

source is independent of the presence of other point sources, may be becomes questionable if the line source has a self generated turbulence such as the present case. The effect of that assumption appears clearly in the large deviation between the results of the two approaches in the second quarter only because the emissions from the traffic road have negligible effect on the calculated concentrations inside the other two quarters.

6 Conclusions

The objective of the present study was to compare the performance of Gaussian plume model approach against a CFD approach through a study for the air pollution problem in a certain urban area located in the central part of Tokyo city.

In conclusion, it can be said that the Gaussian plume model is computationally much more efficient, compared with the CFD simulations regarding the time and modelling effort especially in large scale studies, but unfortunately, it is very poor in predicting the actual concentrations values in cases of high densely built-up areas.

References

- [1] Heinsohn J. & Kabel L., *Sources and Control of Air Pollution*, Prentice Hall: New Jersey, 1999.
- [2] Sofuoglu S., Dispersion from a point source (lectures on air pollution). Department of Chemical Engineering: Izmir Institute of Technology, 2002.
- [3] Griffiths F., Errors in the use of the Briggs parameterization for atmospheric dispersion coefficients. *Atmospheric Environment*, **28 (17)**, pp. 2861-2865, 1994.
- [4] Ashoch K. & Patil S., A general finite line source model for vehicular pollution prediction. *Atmospheric Environment*, **23 (3)**, pp. 555-562, 1989.
- [5] Chock P., A simple line source model for dispersion near roadways. *Atmospheric Environment*, **12 (4)**, pp. 823-829, 1977.
- [6] Xiaomin X., Zhen H. & Jia S., Impact of building configuration on air quality in street canyon. *Atmospheric Environment*, **39 (25)**, pp. 4519-4530, 2005.
- [7] Huang H., Ooka R. & Kato S., Urban thermal environment measurements and numerical simulation of an actual complex urban area covering a large district heating and cooling system in summer. *Atmospheric Environment*, **39 (34)**, pp. 6261-6480, 2005.
- [8] STAR-CD, Methodology STAR-CD version 3.2, Computational Dynamics Limited, 2004.
- [9] Murakami S., Mochida A., & Hayashi Y., Modification of production terms in k- ϵ model to remove overestimation of k value around windward corner. *The 10th Wind Engineering Symposium*, 199-201, (in Japanese), 1988.

