Vehicular Pollutants Dispersion Modelling Analysis In Street Canyons Using Computational Fluid Dynamics (CFD)

Abhishek Pratap Singh
Assistant Professor, ITM University, Gurgaon

ABSTRACT:

Street canyons of dense cities have high level of pollutant concentrations in their lower parts especially at the pedestrian levels. Air pollutants from vehicular emissions entrapped by street canyons within their geographical or physical boundaries. In narrow and deep street canyons local receptor densities are high. Therefore, the study of the effect of vehicular pollutants dispersion in urban street canyons is an important and challenging problem. The role of Computational Fluid Dynamics (CFD) has enhanced for accurate evaluation of the flow field in lesser time and for avoiding setting up complex experiments in environmental flows. Such studies will be helpful in improving air quality management strategies and to reduce the potential health risks within the urban street environments. Numerical simulation, using ANSYS CFX -12 code using the various test parameters e.g. wind direction and aspect ratio have been used. The numerical simulation has been done using standard, RNG (Renormalized Group) and realizable k-ε turbulence models. The results of the simulations have been analyzed and further compared with those obtained from numerical modeling using Computational Fluid Dynamics (CFD) code of other researchers.

Key words: Street Canyon, Pollution concentrations, Computational Fluid Dynamics, *k-ε turbulence models, ANSYS-FLUENT code.*

1. INTRODUCTION:

Sharma and Ray Choudhary (1996) have reported that in India, higher incomes, mobility, expanded cities and greater employment opportunities have increased the demand for motorized transport, resulting in disproportionately high concentrations of vehicles in the urban areas. Air pollution from vehicular emissions is one of the most serious and rapidly growing problems in various urban centers of the world. Usually, long distances between home and place of work, better incentives for personalized transport, and inadequate / poor quality public transport have further aggravated this problem.

The so-called "canyon effect" is associated with the reduced natural ventilation which often leads to a great health hazard for the exposed population and irreversible damage to the eco-system. Currently, control of air quality inside the street canyons is important as the building complexes on either side of the street induce the flow recirculation resulting in more or less stagnant conditions resulting in pollution hot spots. Automotive emissions take place at the ground level; therefore, they have a direct impact on the human health especially, on the drivers, bicyclists, motorcyclists, pedestrians, people working nearby and passengers traveling in the vehicles. Also, polluted air invariably gets introduced to the indoor air environments of the buildings and office complexes situated on both sides of the roadways. It may be noted that pollution levels are likely to be very high if the buildings on either sides are high and the streets are narrow and in addition, associated with heavy vehicular traffic. From the population exposure point of view, study of air quality in urban street canyons is of paramount importance, since the highest pollution levels and considerably larger environmental impacts have often been observed in these situations.

In air pollution problems, the application of air quality models forms one of the most important components of urban air quality management. The main motivation of the proposed project is to develop a suitable numerical simulation model in the near field of roadways using the CFD approach. An effective air quality management system must not only be able to provide information on the current status but also predict the likely future trends in the region so that the concerned authorities and regulatory agencies can realistically assess the efficacy of various air pollution control management strategies implemented and take 'informed' policy decisions for their perspective planning.

Over the past two decades, significant progress has been made in understanding, measuring and modeling dispersion within urban street canyons. These studies have greatly helped not only in understanding dispersion phenomena, but also in determining the pollutant concentrations within the street canyon as a function of building dimensions, upwind building configuration, wind direction with respect to building configuration and roof geometry (Kastner – Klein et al.2000) Very few studies have been carried out in India to know the effect of vehicle induced mixing on pollution dispersion phenomena in the urban street canyons under heterogeneous traffic conditions (Sharma et al., 2004). Various researchers have successfully used FLUENT software in the recent past for the dispersion modeling in street canyon, (Meroney et al.,2000; Chan et.al., 2003; Vardoulkis, S. et al (2003)

In the present study, an attempt has been made to find out the extent of application of CFD software ANSYS-CFX-12 in the pollution predictions within urban street canyons for the c case of two-dimensional street canyon. Comparison of numerical simulations has been done with the similar case of the wind tunnel test results conducted for an isolated two dimensional street canyon model (Awasthi -2006). The numerical simulation is based on the Reynolds averaged Navier–Stokes equations coupled with a series of standard, realizable and RNG (Renormalized group) k-ε turbulence models.

2. URBAN STREET CANYON CONFIGURATIONS:

The present study reported herein, is for the case of regular type of street canyon with Aspect Ratio (AR) = 1.0, which is the ratio of height (H) of the canyon by the width (W) of the street. It is important parameter. Oke (1988) classified the types of street canyons based on the aspect ratio criteria. A street canyon is termed as regular / uniform, if it has an aspect ratio of approximately 1, with no major opening on the walls. Wide / shallow canyons have the aspect ratio of 0.5 and below. On the other hand, a deep canyon is associated with an aspect ratio of 2 or higher. Geometry of a street canyon is shown in Fig. 1.

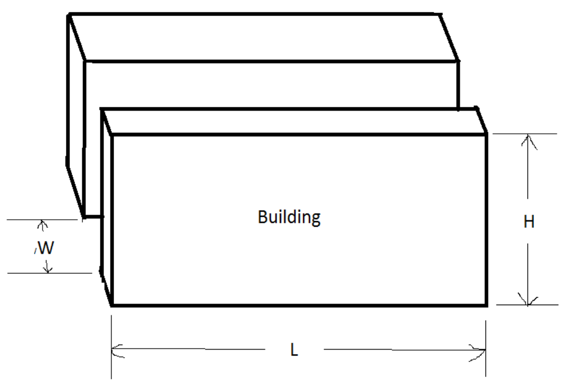

Fig. 1: A typical configuration of a Street Canyon.

Urban street canyons may also be classified as symmetric (or even) canyons, if the buildings flanking both sides of the street have approximately same height, or asymmetric, if there are significant differences in building heights on both sides of the street canyon. Asymmetric canyons with high rise buildings in downward direction are termed as step up canyons and reversibly these are called step down canyons. The upward side of the canyon is called leeward and downward is windward when the wind flow is perpendicular to street canyon. Length (L) of canyon usually represents the road distance between two major intersections. Based on the L/H ratio, street canyons may, further be classified into short (L/H=3), and long canyons (L/H=7). When the wind blows in the direction perpendicular to the longitudinal axis of street canyon, a clockwise vortex circulation is generated. This may be caused due to the transfer of momentum across the shear layer of the roof height. In this situation, the bulk of the flow does not enter into the street canyon and the flow inside the street canyons forms a vertical type f flow. Oke (1988) has reported that there are basically three flow regimes in the street canyons in terms of street canyon width W, and building height H, *Skimming Flow (SF)* with a characteristic vortex being built up for a relatively small aspect ratio (W/H <1.538), *Wake interference Flow (WIF)* for an intermediate aspect ratio (1.538 < W/H< 2.5), *Isolated Roughness Flow (IRF)* for a large aspect ratio (W/H>2.5).

3. *COMPUTATIONAL MODELING:*

To analyze the pollutant transport in street canyons, numerical modeling is commonly adopted because of its low cost and full control of the parameters. In numerical modeling, computer based numerical methods are used to analyze the systems that utilize CFD coupled with a turbulence model. Currently, CFD has found its applications in a vast range of industrial and environmental fields. Various types of numerical models have been employed to simulate urban flow and pollutant dispersion. CFD has been widely used in solving complex problems of dilution / dispersion phenomenon of Environmental Fluid Mechanics mainly due to the following reasons:

CFD modeling is based on the numerical simulation of the governing fluid flow and dispersion equations such as, Mass Conservation (Continuity) Equations, Momentum Conservation (Navier - Stokes) Equation in x, y, and z directions and Transport Equation for pollutant Concentration, Which are derived from basic conservation and transport principles.

The SIMPLE algorithm (Semi-Implicit Pressure Linked Equations) has been used in the CFX -12. Numerical predictions depend primarily on the sub-division of the region of interest into grid cells, choice of turbulence model, accuracy of discretization scheme and choice of boundary condition. There are three key elements in CFD that help in solving any type of flow problems: Turbulence Modeling, Grid generation and Algorithm Development

Two-equation k-ε turbulence model is the most popular model among various kinds of approaches for modeling Reynolds stresses. RANS approach has been widely used for flow and dispersion calculations in complex environments. If the mean flow is steady, the governing equations will not contain derivatives and steady state solutions can be obtained economically.

The renormalization group k-ε model based on RNG theory and realizable k-ε turbulence model are the improved versions to the standard k-ε turbulence model. The main idea of the RNG model lies in the systematic removal of small scales of turbulence by representing their effects in terms of larger scale motion and a modified viscosity.

RNG based k-ε model are the same as those with the standard k-ε model for incompressible, isothermal, and non reacting flows, but it provides an analytically derived differential formula for determining the effective viscosity and an additional term for the ε equation. In addition, the RNG based k-ε model also provides an analytical formula for determining the turbulent Prandtl number instead of a constant value for the standard k-ε model.

4. NUMERICAL SIMULATION OF VEHICULAR POLLUTANTS DISPERSION IN STREET CANYONS:

The flows over the street canyon model have been solved using steady-state Reynolds-averaged Navier-Stokes (RANS) equations. Sini et al. (1996), the air within the street canyon can be regarded as an incompressible turbulent inert flow, and the air and pollutant densities have been assumed to be constant. These assumptions are reasonable for lower most atmosphere environment. Thermal effect in the street canyon is not taken into consideration in the present study due to this the turbulence production due to the buoyancy effect has not been included.

5. RESULTS AND DISCUSSIONS:

Fig. 2 shows the velocity vector diagram for perpendicular wind direction (θ = 90^0) at the mid section of the street canyon for A.R. = 1, using RNG, k - ε model. This represents computed values of velocities of the typical skimming type of flow in the street canyon.

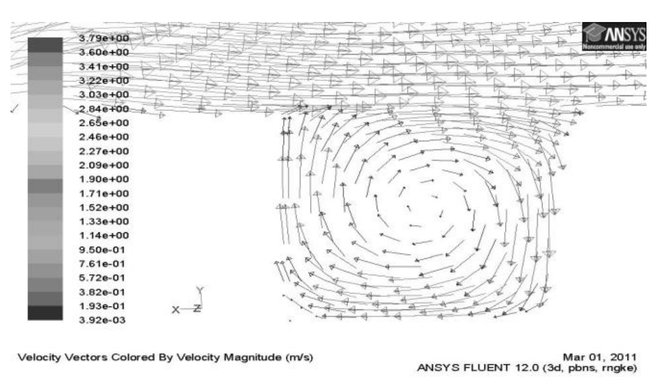

Fig. 2 : diagram showing the velocity vectors with in a Street Canyon for the case of AR = 1. 0

This figure shows that a small part of the high velocity main streams flow enters the windward side and forms a vortical pattern. Due to the high kinetic energy content of the main flow, there is relatively stronger mixing and dilution of pollutants in the windward side and then flow moves towards the leeward side of the canyon. The flow velocities are considerably reduced as can be visualized by relatively larger spacing between the stream lines in the leeward side. Due to this, relatively lesser amount of pollutant scavenged in the leeward side.

The non- dimensionalized pollution level concentration parameter K is defined by the following equation:

$$K = C \,.U_{ref} \,.\, L \,.\, H \,/\, (Q) \quad (1)$$

where, K = Normalized concentration(ppm)

C_0 = Measured concentration(vol/vol)

Uref = Free stream velocity (m/s)

L = Length of line source (m)

H = Characteristic height of the building (m) and

Q = Strength of the tracer gas emission (m^3/s)

The non-dimensionlized pollution parameter K has been evaluated for perpendicular wind direction (θ = 90^0), using the Fluent software using three k-ε turbulence models. They are namely: Standard, Renormalization Group (RNG) and Realizable. The numerically obtained results using these models have been compared with experimentally obtained values of K at different heights of street canyon by physical modeling study using Environmental Wind Tunnel (EWT), which has been reported by Awasthi and Chaudhry (2006). These have been shown in Fig. 3.

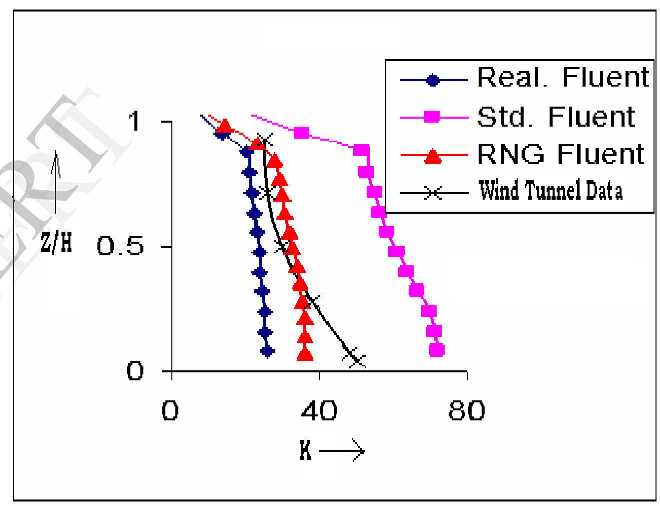

Fig. 3: The comparison of numerically obtained pollution levels at different heights on the Leeward wall of the street canyon of AR =1, with the experimental values of Awasthi (2006).

It is obvious from the Fig. 3, that normalized pollution concentration values obtained using the RNG k-ε model are closer to the experimentally determined values of Awasthi (2006). Both Realizable and

Standard, k-ε models, generally under predicts and over predict respectively the experimental data of the physical modeling study of EWT mentioned above as well as other complex type of flows.

6. CONCLUDING REMARKS;

The following broad conclusions have been arrived in the present study:

1. The present study indicates that fluent code can predict the flow field in urban street canyon. In other words, CFD code FLUENT has also enabled to reproduce accurately the qualitative features of airflow like separation/recirculation areas.
2. The shape of the main vortex is distorted and shifted towards the wind ward side. The airflow faster when it approaches the wall of the building and the ground level and slow in the centre of vortex.
3. In the present study, it has enabled to provide more realistic information on the dispersion of pollutants over an urban street canyon for AR = 1. CFD code, FLUENT, can predict many of the wind tunnel results for average flow field and concentrations by choosing appropriate boundary condition, grid resolution and turbulence model.
4. At perpendicular wind direction ($\theta = 90^0$), it has been observed that the FLUENT computes higher concentrations on leeward wall as compared to the windward wall.
5. The pattern of normalized variation of RNG model has been found to be the best matched with the results of Awasthi (2006) experimental study. In other words, RNG model works best when compared to the standard and realizable k-ε turbulence models for case Street canyon concentration predictions as well as generally for other complex type of flows.
6. CFD is a very powerful modeling technique in the analysis of air pollution scenarios considered herein. It is low cost as compared to the full scale experimental study. Further, it has been observed that

7. REFERENCES:

8.

9. [1] Seema Awasthi, Physical modeling and Numerical Simulation Study of Line Sources Dispersion in Urban Street Canyon,Department of Applied Mechanics, Ph D Thesis, IIT-D,2006, pp 1-188.

10. [2] Robert N. Meroney, Michel Pavageau, Stilianos Rafailidis, Michael Schatzmann , Study of line source characteristics for 2-D physical modeling of pollutant dispersion in street canyons, Journal of Wind Engineering and Industrial Aerodynamics, Volume 62, August 1996, pp 37-56.

11. [3] Jae-Jin Kim, Jong-Jin Baik, A numerical study of the effects of ambient wind direction on flow and dispersion in urban street canyon using the RNG *k*–ε turbulence model, Atmospheric

Environment, Volume 38, June 2004, pp 3039-3048.

12. [4] T. L. Chan, G. Dong, C. W. Leung, C. S. Cheung, W. T. Hung Validation of a two-dimensional pollutant dispersion model in an isolated street canyon, Atmospheric Environment, Volume 36, Issue 5, February 2002, pp 861-872.

13. [5] F.J. Jime´nez-Horrneroa, J.V. Gira´ldezb, E. Gutie´rrez de Rave´c, F.J. Moral, Description of pollutant dispersion in an urban street canyon using a two-dimensional lattice model, Atmospheric Environment, Volume 41, Issue 5,2007, pp 221-226.

14. [6] N.A. Mazzeo, L.E. Venegas and P.B. Martin, Analysis of Full Scale Data Obtain in a Street Canyon, Depatmento de Ciencias ded la atmosfera y los oceanos, Volume 20, 2007, pp 93-110.

15. [7] Pei Shui, Chun-Ho- Liu and Yuguo Li, CFD Analysis of Pollutant Removal Mechanism In Urban Street Canyons, Department of Mechanical Engineering, The University of Hong Kong, Volume 31, July 2009, pp 137-140.

16. [8] Jong-Jin Baik, Yoon-So kang, Jae-Jin kin, Modeling Reactive Pollutant Dispersion in an urban Street Canyon, Atmospheric Environment, Volume 41, 2007, pp 934-949.

17. [9] T.L Chan, g Dong, C.W. Leung, C.S. Cheung, W.T. Hung, Validation of a two-dimensional pollutant dispersion model in an isolated street canyon, Atmospheric Environment, Volume 36, 2002, pp 861-872.

18. [10] Federica Santese et. al., CFD Simulation for the Study of Thermal Effects on Flow and Pollutant Dispersion in Urban Geometries, Department of Engineering, University of Cambridge (11th International conference on harmonization within Atmospheric Dispersion Modeling for Regulatory Purposes)

19. [11] K. Ahmad, M. Khare, K.K. Chaudhry, Wind tunnel simulation studies on dispersion at urban street canyons and intersections, Journal of Wind Engineering and Industrial Aerodynamics, Volume 93, Issue 9, September 2005, pp 697-717.

20. [12] P. Salizzoni, L. Soulhac, P. Mejean, Street canyon ventilation and atmospheric turbulence, Atmospheric Environment, Volume 43, Issue 32, October 2009, Pages 5056-5067.

21. [13] Baik, J.-J., Kim, J.-J., Fernando, H.J.S, A CFD model for simulating urban flow and dispersion, Journal of Applied Meteorology, Volume 42, 2003, pp 1636–1648.

22. [14] P. Kastner-Klein, E. J. Plate, Wind-tunnel study of concentration fields in street canyons, Atmospheric Environment, Volume 33, Issues 24-25, October 1999, Pages 3973-3979.

23. [15] Sharma, Physical simulation of vehicular pollution dispersion an an isolated street canyon under heterogeneous traffic conditions, Ph D Thesis, IIT Delhi, 2005.

24. [16] John D. Anderson, Computational Fluid Dynamics- The basic Applications, Department of Aerospace Engineering, University of Maryland, pp 1-167.

25. [17] Baik, J-J. Kim, On the Escape of Pollutants from Urban Street Canyon, Atmospheric Environment, Volume 36, 2002, pp527-536.

26. [18] FLUENT 12.1 User's Guide, Volume 1-7, Fluent Inc., Lebanon, 2009.

27. [19] Central Road Research Institute, (CRRI).

28. [20] Indian Meteorological Departments of India (IMD).

29. [21] Sini et. al., Pollutant Dispersion and Thermal effects in Urban Street canyons, Atmospheric environment, Volume 30, pp 2659-2677.