The Nucleus **53**, No. 3 (2016) 214-220

www.thenucleuspak.org.pk

The Nucleus

ISSN 0029-5698 (Print)

ISSN 2306-6539 (Online)

Prediction of Comfort Parameters for Naturally Ventilated Underground Car Parks

S. Khatoon[1*] and A.N. Shah

[1]*Department of Mechanical Engineering, University of Engineering & Technology, Lahore, Pakistan*

[2]*Department of Mechanical Engineering, University of Engineering & Technology, Lahore, Pakistan*

saboorakhatoon@yahoo.com; anaeems@uet.edu.pk

ARTICLE INFO

Article history :

Received : 12 February, 2016

Revised : 09 April, 2016

Accepted : 21 September, 2016

Keywords :

Car parking

Natural ventilation system

Air Changes per Hour

Carbon mono oxide generation rate

FLUENT

ABSTRACT

With the increased population and rapid motorization in urban areas of Pakistan, the concept of underground car parking areas in multistory buildings is increasing day by day and is a viable solution for modern high rise shopping malls. However, maintaining a thermal comfort level and dilution of vehicle emission for naturally ventilated basements is very challenging since the toxic pollutants cause serious health issues. Computational Fluid dynamics FLUENT is a valuable tool for study of mean air speed, pressure and velocity streams and is supportive in identification of red zones inside the building and is becoming a legal requirement. The research is an effort to implement two of the available FLUENT turbulent models using finite volume technique on an existing two level basement under a multistory building, predicting the best suitable model. The evaluation for each model is performed by developing the 3D model for each level of basement, meshed with equal number of cells and analyzed under same boundary conditions. In addition, an effort is made to investigate the gaseous emission from vehicle exhaust. Indoor air quality is assessed by defining the CO generation rate and respective ACH (Air Changes per Hour) are determined using ANSI/ASHRAE 62-1989 for a maximum concentration of 25ppm and 35ppm. The acceptable indoor air quality specifies a fixed ventilation rate of 7.62 L/s.m2 i.e. 1.5 cfm/ft2 of gross floor area, which means that a ventilation flow of about 11.25 air changes per hour is required for garage with 2.5m (8 ft) ceiling height. Results showed that for level-I the number of air changes required are 17 with a concentration limit of 25ppm of CO in an hour while with limit of 35ppm 12 ACH are enough for dilution of indoor air. For Level-II, which is used to its maximum capacity, the number of air changes required are 11 with limit of 25ppm but drop down to 7 with concentration limit of 35ppm. A parametric analysis for garage height and CO generation shows that with an increase in garage height the ventilation rate required to keep CO level minimum or within acceptable level also increases as the volume of car park increases.

1. Introduction

Keeping in view the latest trends for modern style high rise shopping malls in cities for the parking of vehicles multistory parking are now designed and constructed. In order to make sure proper ventilation system to avoid suffocation and any possible causalities due to vehicles exhaust and smoke due to fire smoke and to provide a comfort zone for multistory underground parking, proper building design for intake and exhaust air is considered important. Air quality, as well as thermal comfort level also needs attention. In short, justifiable environment is a documented need in building industry. Keeping in view its importance; proper computational fluid dynamics analysis is now made a compulsory element to validate the design for velocity and pressure streams to identify any red zones and is becoming a legal requirement.

For the past few decades, researches are being made to dilute the air inside the car parking area. For this purpose, scale down models and office buildings were considered earlier. Computational Fluid Dynamics uses turbulent modeling to quantify the velocity magnitudes and pressure values for any building. Fisk [1] defined two effectiveness parameters for air exchange. These parameters show the nature of air flow pattern inside a building, dislocation of flow in a room, air diffusion effectiveness, mingling, the level of short circuiting, and displacement flow in a room as well as normalized local age of air; this parameter compares the average ventilation rate with ventilation rate locally. Several measurement issues and difficulties were discussed by considering office buildings. Jiang [2] investigated natural ventilation using Steady Reynolds Averaged Navier-Stokes equation (RANS) modeling. The same process of ventilation was also studied using unsteady RANS modeling as well as large eddy simulations (LES). The results showed that the most important factor in determination of ventilation rate is fluctuating flow field for wind driven natural ventilation only. Blocker [3] gave

* Corresponding author

suggestions to improve CFD simulation by concentrating on the simulation of a horizontally regular Atmospheric Boundary Layer over uniformly uneven flat ground addressing wall function problems. Becker [4] addressed thermal comfort and energy conscious design for school buildings. Air-conditioners are used commonly for thermal comfort but they do not fix air quality. Also uncontrolled energy losses occur because of natural ventilation. The proposed scheme of improved ventilation resulted in energy savings of 28-30% for class rooms in a well-made energy conscious building.

Van Hooff [5] discussed a coupled CFD modeling approach considering indoor natural ventilation and urban wind flow. A parametric analysis for geometrically intricate Amsterdam Arena stadium is performed keeping in view the natural ventilation in Netherlands. Krarti [6] studied the requirements for enclosed car parking ventilation considering emission rates of different pollutants emitting from different vehicles and their corresponding effect on human health and environment. Wong [7] conducted a comparative survey for different car parking areas including enclosed car parking in city of Hong Kong through seventy two canister samples for carbon monoxide (CO), temperature and carbon dioxide (CO_2) for indoor and outdoor air quality. Ho [8] achieved a multivariable correlation with engine operating time fraction and ambient temperature by considering a medium-size underground car park investigating the indoor air quality and thermal comfort. The total number of car lots available and six days data for the engine operating time and traffic volume were studied resulting useful predictions for building ventilation system designers.

Ryu [9] investigated the indoor air quality by evaluating the ventilation system using ANSYS CFX. Air flow inside the tunnel was studied by different train running conditions. Results showed that tunnel has high carbon dioxide concentration and air quality reduces as number of train increase. Mohammed [10] evaluated a multi-storey car parking in Kuala Lumpur. The exhaust of automobile vehicles contains harmful toxics including sulphure dioxide, nitrogen oxide, farmeldehyde, carbon mono and dioxide.

Eshack [11] analyzed the CO contours using CFD modeling and multi-zone modeling. The results were compared with experimental data. CFD modeling gave precise distribution while multi-zone modeling gave estimated macroscopic solution. The proper removal of pollutants is important to consider for the health of people who work there. Khalil [12] investigated the Tahreer square; a multi-story car park using six different running conditions and evaluated it against Egyptian code of garages. Numerical simulation was performed using ANSYS FLUENT CFD package. Thermal comfort level was studied according to ASHRAE 55.

In this paper, a study is performed to predict the best suitable FLUENT model for the evaluation of any closed area more specifically enclosed car parks and air ventilation quality with thermal comfort and CO generation rate is assessed. For this purpose the two level basement of a university building in Lahore-Pakistan is considered.

2. Methodology

An ephemeral description of the site under examination, the methodology used for modeling and simulation of velocity and pressure contours, thermal comfort and CO generation rate is described in this section. A brief description of the software package employed for modeling is also included.

2.1 Site Description

The measurement subject is a two level car parking basement of a university building in Lahore, Pakistan. The floor area is 6074 m2 and 6635 m2 for level one and two, respectively. The capacity of cars for level one is 200 and some offices are also available at this level. Level two has capacity of 350 cars. The garage height of each level is 2.4 m. Both levels are naturally ventilated. [Fig. 1 & Fig. 2]

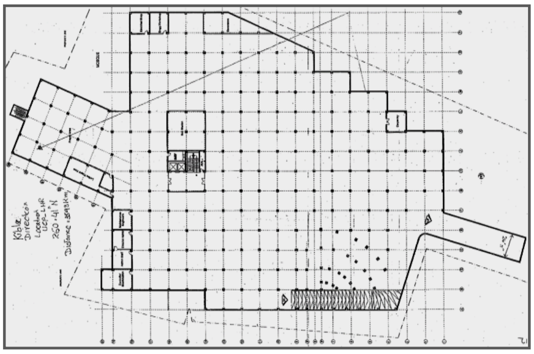

Fig. 1: Plan of Level-I car parking

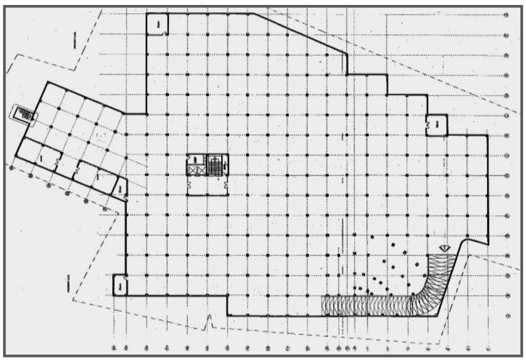

Fig. 2: Plan of Level-II car parking

2.2 Indoor Air Quality monitoring

An enriched indoor air quality is the basic demand for healthy environment inside the basement. Elimination and control of all the airborne pollutants is unachievable; however the contaminates can be controlled within a prescribed limit. A general outline is described below;

a. Walk-Through Survey: In this survey, information about the physical layout of the building is studied and plan of each level is collected. The air inlet and outlet positions along with vents are identified. Average indoor air temperature is measured.

b. Vehicle usage pattern: The total number of vehicles entering and exiting the facility are collected. Type of vehicle, time of vehicle movement in car parking area and typical vehicle paths are collected through discussion with security officer and direct observation.

c. CO generation rate: The CO generation rate is calculated using ASHRAE (945-RP). The number of air changes per hour, considering the generation rate with maximum 25ppm and 35ppm are evaluated. The factors effecting CO generation are age of car, level of car maintenance and type of vehicle.

2.3 Simulation

ANSYS FLUENT (14.0), a commercial CFD [13] code is used for modeling and simulation. CFD is a multidisciplinary tool that involves the concepts of fluid dynamic, analytical mathematics and computer codes. It gives a vision to flow patterns of the fluid which are difficult to predict experimentally and costly to perform. Using the finite volume method, two of the available models; SpalartAllmaras and k-Є are used to simulate the air flow and their results are compared graphically. Basically, FLUENT divides the solution domain into millions of cells and nodes. Numerical techniques are used to compute the solution and problem variables are stored at each node.

Spalart-Allmaras model [14-15] is one equation model. This model is relatively simpler in solving kinematic eddy viscosity for a modeled transport equation. Spalart-Allmaras is useful for adverse pressure gradients in wall-bounded flows but it gives large error in jet flows since it is not calibrated for free shear flows.The original form of Spalart-Allmaras shows that it is effectively a low-Reynolds-number model.

The one-equation model has the following transport equation,

$$\frac{\partial}{\partial t}(\rho\dot{\upsilon}) + \frac{\partial}{\partial x_i}(\rho\dot{\upsilon}u_i) = G_v + \frac{1}{\sigma_{\dot{\upsilon}}}\left[\frac{\partial}{\partial x_j}\left\{(\mu + \rho\dot{\upsilon})\frac{\partial\dot{\upsilon}}{\partial x_j}\right\} + C_{b_2}\rho\left(\frac{\partial\dot{\upsilon}}{\partial x_j}\right)^2\right] - Y_v + S_{\dot{\upsilon}} \quad (1)$$

Where, ρ is the fluid density, $\dot{\upsilon}$ is the transported variable, u_i is the x-direction velocity, G_v is the production of turbulent viscosity, $\sigma_{\dot{\upsilon}}$ and C_{b_2}re constants and v is the molecular kinematic viscosity, molecular dynamic viscosity, $Y_v$is the destruction of turbulent viscosity that occurs in the near-wall region due to wall blocking and viscous damping. $S_{\dot{\upsilon}}$ is the user defined source term. While carrying this research project default values for the constants being used in FLUENT are applied, the model constants have the following constant values; $C_{b_1} = 0.1355$, $C_{b_2} = 0.622$ and $\sigma_{\dot{\upsilon}} = \frac{2}{3}$.

Table 1: Meshing details

Level-I	
Mesh element size	0.5
Number of nodes	424575
Number of Elements	2204765
Minimum volume	1.962705 e-08 m3
Maximum volume	5.559371 e-01 m3
Orthogonal Quality	1.91498 e-01
Aspect ratio	2.5427 e+01
Level-II	
Mesh Element size	0.5
Number of nodes	3006923
Number of elements	15796361
Minimum volume	1.673882 e-05 m3
Maximum volume	4.189613 e-02 m3
orthogonal quality	1.68746 e-01
Aspect ratio	2.4566 e+01

K-Є model [14-15] is the most widely used industrial CFD two-equation turbulence model. It works by cracking the two transport equations and modeling the Reynolds Stresses using the Eddy Viscosity methodology. The Standard k-ε method is capable of solving practical engineering flow calculations and has become a workhorse in the time.

In this model, following transport equation is used;

$$\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_i}(\rho k u_i) = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right] + G_k + G_b - \rho\epsilon - Y_M + S_k \quad \text{(Eq. 2)}$$

$$\frac{\partial}{\partial t}(\rho\epsilon) + \frac{\partial}{\partial x_i}(\rho\epsilon u_i) = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_\epsilon}\right)\frac{\partial\epsilon}{\partial x_j}\right] + C_{1\epsilon}\frac{\epsilon}{k}(G_k + C3\epsilon Gb - C2\epsilon\rho\epsilon 2k + S\epsilon \quad \text{(Eq. 3)}$$

Where,G_k is Generation of turbulence kinetic energy due to mean velocity gradients, $G_b$is Generation of turbulence kinetic energy due to buoyancy, Y_M is Contribution of the fluctuating dilation in compressible turbulence and $C_{1\epsilon}$, $C_{2\epsilon}$, $C_{3\epsilon}$ are constants. σ_k&σ_ϵ are turbulent Prandtl numbers for k and Є, respectively. While considering k-Є model, the default values of constants $C_{1\epsilon}$, $C_{2\epsilon}$, $C_{3\epsilon}$ that are 1.44, 1.92, 0.09 were used. The turbulent prandtl numbers 1.0 and 1.3 were used for k and Є, respectively.

Steps involved in CFD modeling: Three dimensional model is developed in ProE (Creo 12.0) and imported to ICEM CFD 14.0 as .stp file for meshing (Fig. 3 & Fig. 4). Details are provided in Table 1.

Pressure based solver with SIMPLE algorithm is taken for pressure velocity coupling. Inlet air velocity of 0.15m/s with density 1.225 kg/m^3 and viscosity 1.7894 e-5 kg/m-s is assigned at inlets at atmospheric pressure and temperature.

3. Results and Discussion

3.1 Thermal Comfort for Natural Ventilation

It is evaluated based on indoor air temperature criteria for natural ventilation [17]. As shown in Fig. 5: Climate report (annual) Lahore maximum and average temperature and humidity are observed, this is compared with the experimentally determined indoor air

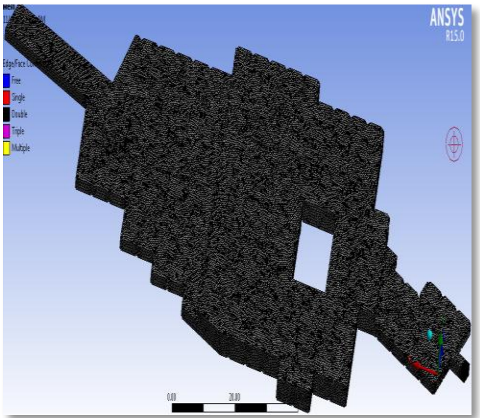

Fig. 3: Meshed volume for level-I

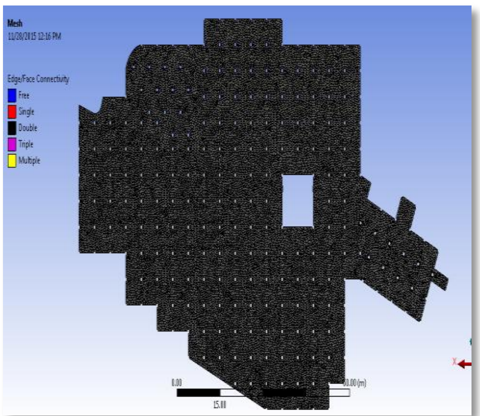

Fig. 4: Meshed volume for Level-II

temperature for each level. Lahore is located between 31°15’-31°45’ N and 74°01’-74°39’E and weather conditions comprise of extremely hot and long summer (month of May, June and July) when temperature ranges up to 40-48°C; with monsoon and dust storm. Fig. 6 shows the comparative evaluation of indoor air temperature for the building and the thermally acceptable indoor temperature described by Humpherys, De Dear and Yang [17].

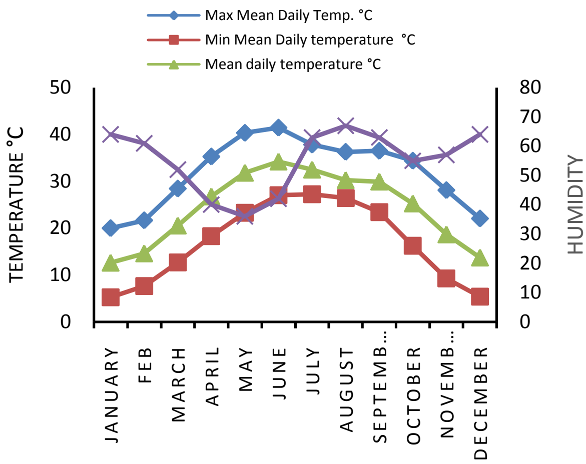

Fig. 5: Climate report (annual) Lahore

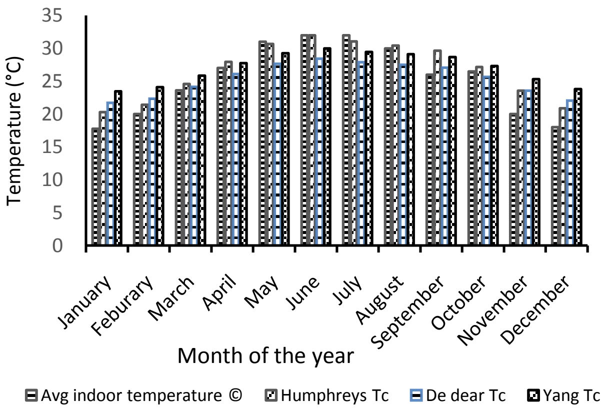

Fig. 6: Comfortable indoor air temperature

(ii). Air flow network: Air flow pattern was studied using FLUENT. The contours developed for velocity for level-I are shown in Fig. 7 & Fig. 8. The velocity magnitudes calculated from each model are compared in Fig. 9. For Level-II, the velocity contours developed are shown in Fig. 10&Fig. 11 and respective comparison of magnitudes is shown in Fig. 12. As per literature survey, the velocity magnitude for air should be in between 0.1-0.2 m/s. The results for Level-I are satisfactory but for level-II, the areas highlighted in contours with values below 0.06 m/s need attention.

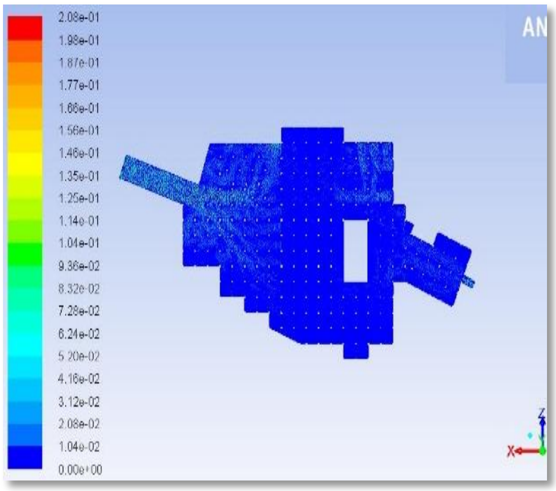

Fig. 7: Velocity contours for level-i using splarat-allmaras

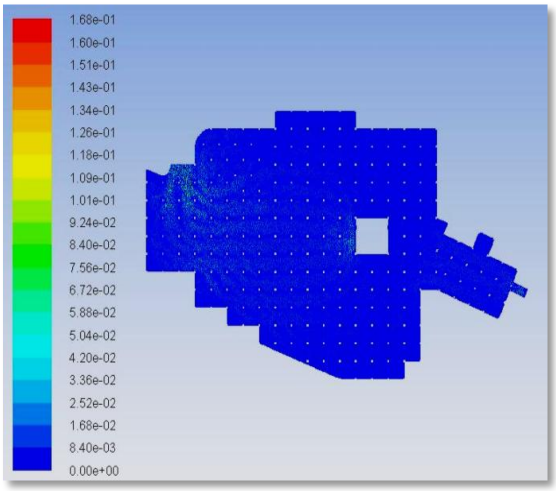

Fig. 10: Velocity contours for level-ii using splarat-allmaras

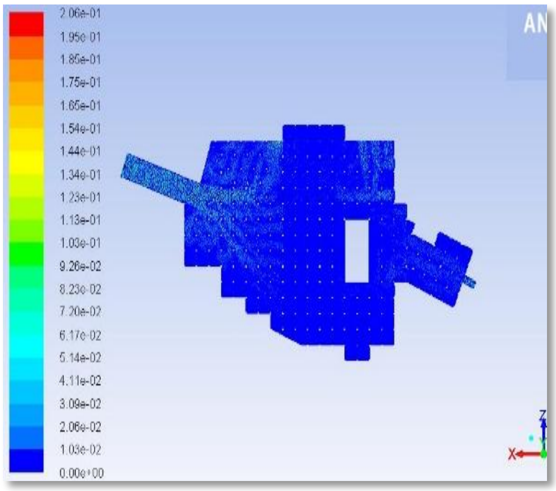

Fig. 8: Velocity contours for level-I using k-e

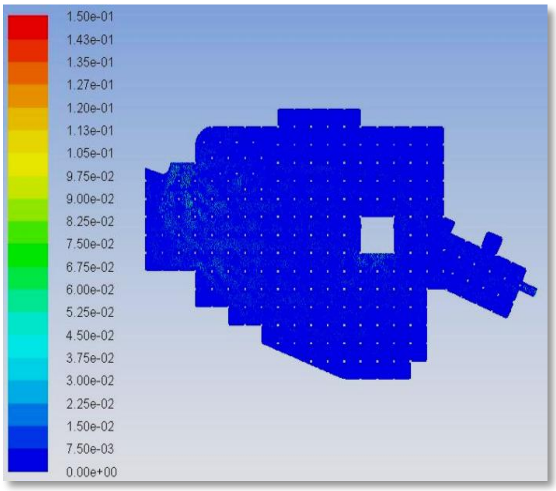

Fig. 11: Velocity contours for Level-II using k-e

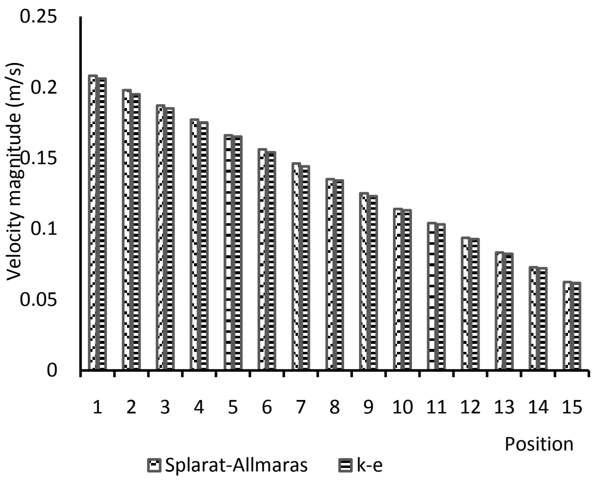

Fig. 9: Velocity magnitude inside level-I

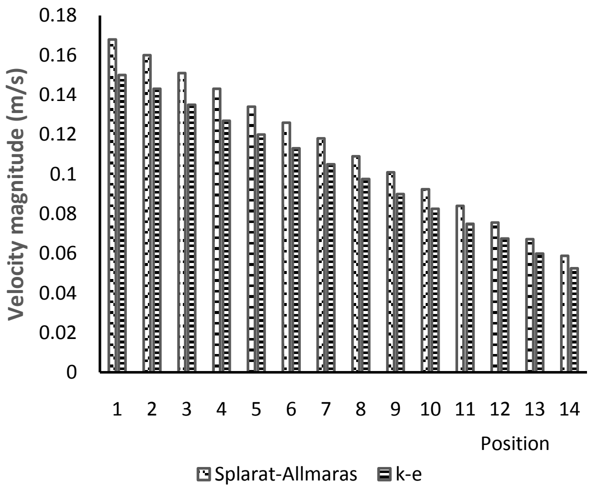

Fig. 12: Velocity magnitude inside level-II

*(iii). Indoor air quality assessment:*It was performed for naturally ventilated car parking basement by monitoring the car movement in each level for a period of 9 hours. Knowing the number of engines in operation at each hour, the CO generation rate was calculated using improved method of Moncef Krarti in ASHRAE 62-1989 for cold emission. CO generation rate in $gr/hr.m^2$ calculated for each level is shown in Fig. 13&Fig. 14. According to ANSI/ASHRAE Standard 62-1989, *Ventilation for acceptable indoor air quality* specifies a fixed ventilation rate of below 7.62 $L/s.m^2$i.e 1.5 cfm/ft^2 of gross floor area, which means that a ventilation flow of about 11.25 air changes per hour is required for garage with 2.5m (8 ft) ceiling height.

From the study of Noreen [18], the CO level at the site of this building specifies a concentration of 25ppm to 35ppm. Therefore air changes per hour (ACH) with CO generation rate for each concentration can be determined. Fig. 15 shows that for level-I the number of air changes required are 17 with a concentration limit of 25ppm of CO in an hour while with limit of 35 ppm 12 ACH are enough for dilution of indoor air. For Level-II, which is used to its maximum capacity, the number of air changes required are 11 with limit of 25ppm but drop down to 7 with concentration limit of 35ppmas in Fig. 16. [11]A parametric analysis for garage height and CO generation shows that with an increase in garage height the ventilation rate required to keep CO level minimum or within acceptable level also increases as the volume of car park increases. The optimized ventilation rate can easily be calculated by;

$$v = v_{ref} \cdot \left(\frac{h}{h_{ref}}\right)^3 \quad (4)$$

Where, v is required minimum ventilation rate for garage height of h when v_{ref} is the required minimum ventilation rate for reference garage height of h_{ref}.

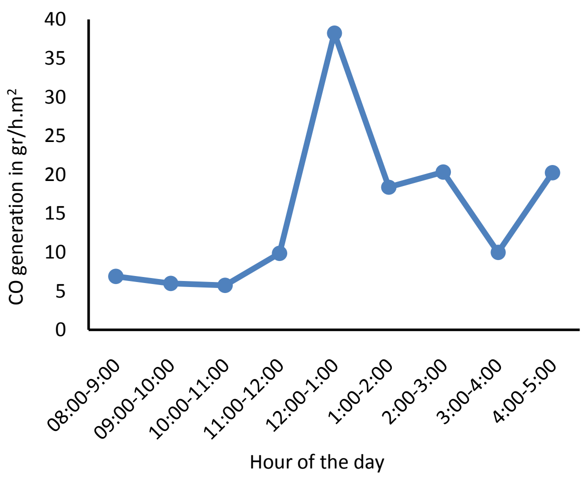

Fig. 13: CO generation rate with respective one hour interval at level-I

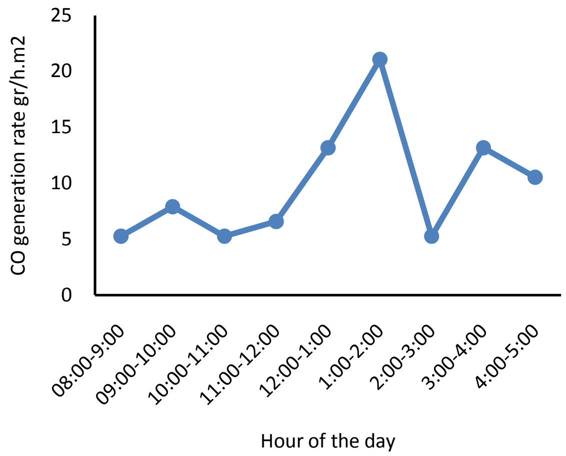

Fig. 14: CO generation rate with respective one hour interval at level-II

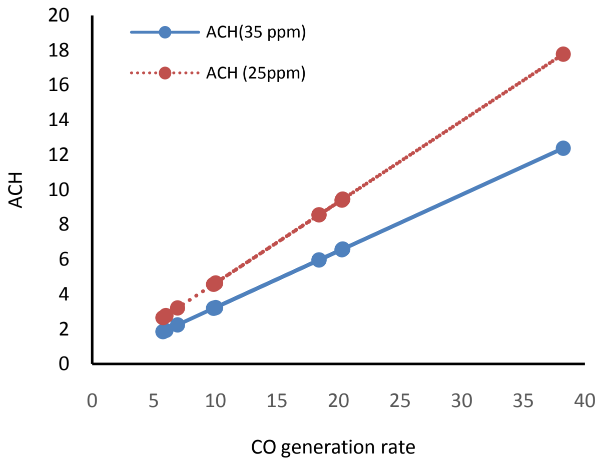

Fig. 15: Air changes per hour with CO generation rate for Level-I

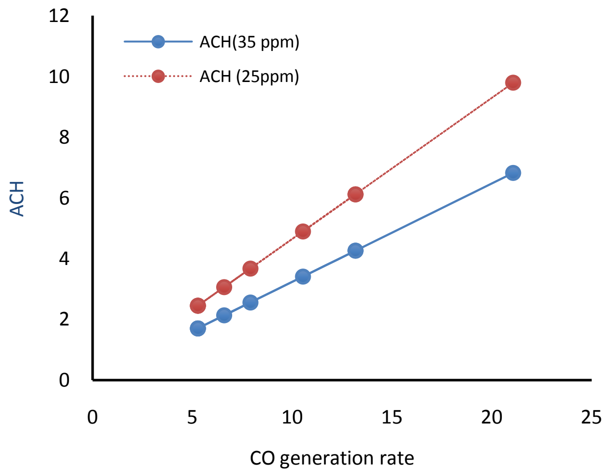

Fig. 16: Air changes per hour with CO generation rate for level-II

4. Conclusion

The present research work focuses on the evaluation of ventilation system of an underground car parking area in Lahore. The car parking area is a two level basement with a capacity of 300 cars per floor approximately. Thermal comfort level is maintained using natural ventilation system. Assessment of natural ventilation system is performed by the evaluation of thermal comfort inside the building bearing in mind the outdoor temperature and climate record. The system was successfully developed and studied in FLUNET. FLUENT uses Finite Volume approach to get the best approximate results and it is good in solving structured as well as unstructured meshes. Three dimensional drawings were generated and simulated on a 4 core computer with 4 GB of ram. Velocity and pressure contours are drawn by means of FLUENT 14.0 with SIMPLE algorithm using Splarat-Allmaras and k-Є model for a 3D developed model. Both the geometries were simulated separately to minimize simulation time consumption. Major part of time was consumed in initial geometry designing. Post processing results showed that the fresh air inlet is set to be sufficient to change the air inside the car parking area with no stagnant areas. Considering three dimensional geometry for each basement level separately, contours and plots drawn for each model showed that two equation model i.e. k-Є model provided the more detailed results as compared to one equation model i.e. Splarat-Almaras model. Indoor air quality is judged by finding the CO generation rate with required number of air changes per hour, for level-I the number of air changes required are 17 with a concentration limit of 25ppm of CO in an hour while with limit of 35ppm 12 ACH are enough for dilution of indoor air. For Level-II, which is used to its maximum capacity, the number of air changes required are 11 with limit of 25ppm but drop down to 7 with concentration limit of 35ppm. As per the criteria of World Health Organization, the maximum concentration of carbon monoxide should be 25ppm, therefore Level-II which is used at its maximum capacity needs increased number of air changes in an hour.

Garage height is an important parameter to maintain the comfort limit for pollutant dilution. But increase in garage height will lead to increased volume which will need increased number of air changes. The subject is a multistory building in operation, any change in dimensions is not possible, and therefore utilization of jet fans is the only option available. Further research can be made to define the number and location of jet fans for Level-II.

Acknowledgement

The authors are grateful to Dr. Gulraiz Ahmed, Dr. Muhammad Rizwan Shad, Dr. Muhammad Abid and Dr. Tariq Manzoor for sharing their pearls of wisdom with us during the course of this research.

References

[1] WJ Fisk and D. Faulkner, “Air exchange effectiveness in office buildings: measurement techniques and results”, Proc. of the 1992 Int. Symp. on Room Air Convection and Ventilation Effectiveness, Jul 2, pp. 22-24, 1992.

[2] Y. Jiang, , C. Allocca and Q. Chen, “Validation of CFD simulations for natural ventilation”, Int. J. of Ventilation, vol. 2, no. 4., pp. 359-369, 2004.

[3] B. Blocker, T. Stathopoulos and J. Carmeliet, "CFD simulation of the atmospheric boundary layer: wall function problems", Atmospheric Environment, vol. 41, no. 2, pp.238-252, 2007.

[4] R. Becker, I. Goldberger and M. Paciuk, “Improving energy performance of school buildings while ensuring”, Building and Environment, vol. 42, no. 2, pp. 3261–3276, 2007.

[5] T. Van Hooff and B. Blocken, "Coupled urban wind flow and indoor natural ventilation modelling on a high-resolution grid: A case study for the Amsterdam ArenA stadium", Environmental Modelling & Software, vol. 25, no. 1, pp.51-65, 2010.

[6] M. Krarti and A. M. Ayari, “Overview of existing regulations for ventilation requirements of enclosed vehicular parking facilities”, ASHRAE Transactions, vol. 105, pp. 18-32, 1999.

[7] Y. C. Wong, D. W. M. Sin and L. L. Yeung, “Assessment of the air quality in indoor car parks”, Indoor and Built Environment, vol. 11, no. 3, pp.134-145, 2002.

[8] J.C. Ho, H. Xue and K.L. Tay, “A field study on determination of carbon monoxide level and thermal environment in an underground car park”, Building and Environment, vol. 39, no 1, pp. 67-75, 2004.

[9] K. J. Ryu, M. Juraeva, S.H. Jeong and D.J. Song, “Ventilation efficiency in the subway environment for the indoor air quality”, Proc. of World Academy of Science, Engg. and Tech., no. 63, 2012.

[10] N. I. Mohammed, N. Othman and K. B. Baharuddin, “Air quality profile in an enclosed car park”, Applied Mechanics and Materials, vol. 567, pp. 3-7, August 2014.

[11] A. Eshack and M. P. Maiya, “Monitoring and simulation of mechanically ventilated underground car parks”, J. of Thermal Engg., vol. 1, no 5, pp.295-302, 2015.

[12] E.E. Khalil, Y. M. Shoukry, H. A. Osman and H. Harridy, “Investigation of ventilation system performance of tahreer car park using CFD”, J. of Energy, vol. 2, no. 2, pp. 81-89, 2015.

[13] A. N. S. Y. S. Fluent, 12.0 Documentation, Ansys Inc. 2009.

[14] N. A. Najar, D. Dandotiya and F. A. Najar, “Comparative analysis of k-ε and spalart-allmaras turbulence models for compressible flow through a convergent-divergent nozzle”, The Int. J. of Engg. and Sci. (IJES), vol. 2, pp.8-17, 2013.

[15] A.N.S.Y.S. Fluent, 14.0 Tutorial Guide–ANSYS Inc., Canonsburg, PA., 2011.

[16] H. Montazeri and B. Blocken, “CFD simulation of wind-induced pressure coefficients on buildings with and without balconies: validation and sensitivity analysis”, Build. and Environ., vol. 60, pp.137-149, 2013.

[17] X. Su, X. Zhang and J. Gao, “Evaluation method of natural ventilation system based on thermal comfort in China”, Energy and Buildings, vol. 41, no 1, pp.67-70, 2009.

[18] N. Ashraf, M. Mushtaq, B. Sultana, M. Iqbal, I. Ullah and S.A. Shahid, “Preliminary monitoring of tropospheric air quality of Lahore City in Pakistan”, Sustainable Development, vol. 3, no 1., pp.19-28, 2013.