ISBN: 978-628-96613-1-6. ISSN: 2414-6390. Digital Object Identifier: https://dx.doi.org/10.18687/LACCEI2025.1.1.2111

Optimization of Ventilation in Multi-Family Parking Lots Using Computational Fluid Dynamics for the Reduction of Carbon Monoxide

David Janampa[1]; Miguel Aldunate[1]
[1] Universidad Tecnológica Perú (UTP), Perú, *u17200509@utp.edu.pe, u17200028@utp.edu.pe*

***Abstract*– Urban growth has led to an increase in the construction of multi-family buildings, which has increased the demand for underground parking spaces. In this study, the ventilation system of these spaces was optimized using Computational Fluid Dynamics (CFD) simulation to reduce the carbon monoxide concentration. The mathematical modeling of air flow and carbon monoxide dispersion was based on Navier-Stokes equations, which allowed the evaluation of different ventilation configurations and improved air distribution. The results show that the optimization reduced the carbon monoxide evacuation time from 13.20 minutes to 7.00 minutes, increased the air velocity from 0.1 m/s to 1.5 m/s, and reduced energy consumption by 40%, ensuring compliance with the American Society of Heating, Refrigerating and Air-Conditioning Engineers (ASHRAE) 62.1 standard. As a result, these results can be used to design and improve underground parking garage ventilation systems.**

***Keywords*-- Computational Fluid Dynamics (CFD), Carbon Monoxide (CO) Reduction, Underground Parking Ventilation, Airflow Optimization, Energy Efficiency.**

I. Introduction

Urban growth has stimulated the construction of multi-family buildings in metropolitan areas, which has led to an increase in the demand for underground parking due to the scarcity of surface parking. At the same time, the growing number of vehicles has increased the need for adequate infrastructure to protect vehicles, especially in high-density buildings [1].

Although underground garages are an efficient solution in terms of space utilization, they also pose a significant risk to air quality due to the accumulation of carbon monoxide (CO). This toxic gas, a product of incomplete combustion in gasoline and diesel engines, is colorless, odorless and difficult to detect without a proper monitoring system [2]. Prolonged exposure can cause symptoms such as dizziness, nausea, confusion, and even death at high concentrations [3]. For this reason, regulatory agencies have established acceptable concentration limits and criteria for the design of ventilation systems in these spaces.

To mitigate these risks, it is essential to implement efficient ventilation systems that ensure the removal and dilution of CO, prevent its accumulation, and maintain adequate indoor air quality. Various regulations, such as the National Building Code of Peru (RNE-EM030-2020) and ASHRAE standards, establish specific parameters for the design of these systems with the aim of keeping CO concentrations within acceptable limits [4]. However, conventional ventilation systems may not be efficient enough in distributing air flow, which can result in areas of pollutant accumulation and prolonged evacuation times.

Conventional ventilation systems in underground parking garages have several limitations that affect their efficiency. First, Peruvian regulations set minimum requirements for air extraction, but do not provide for specific strategies to optimally distribute airflow throughout the space, which can lead to the formation of areas with high CO concentrations [5]. In addition, many systems use extractors uniformly, without considering the variability of CO generation depending on the distribution of vehicles and the natural ventilation conditions of the parking lot [6].

A viable alternative to improve the efficiency of these systems is Computational Fluid Dynamics (CFD) simulation, which allows the modeling of airflow behavior and CO dispersion within the parking garage. Thanks to these simulations, it is possible to evaluate different ventilation configurations and determine the most effective strategy to ensure rapid and uniform CO removal [7].

The research gap lies in the lack of applied studies in the Peruvian context that integrate CFD simulation tools to optimize the design of ventilation in underground car parks. This study aims to fill this gap by providing a detailed analysis of the effectiveness of different ventilation approaches and proposing a solution based on computational simulations that guarantee compliance with current regulations and the safety of the occupants.

CFD simulation has been used in several studies to evaluate and improve the ventilation of underground car parks. Previous research has shown that the use of CFD allows the modeling of pollutant dispersion and the evaluation of the effectiveness of different ventilation configurations prior to their physical implementation [8].

For example, in [9] they compared natural and mechanical ventilation in underground car parks and showed that mechanical ventilation significantly improves air quality by optimizing airflow distribution. On the other hand, in [10] they proposed architectural strategies that would promote natural ventilation to reduce the dependence on mechanical systems, while in [11] they used CFD to validate their ventilation design in a multi-family parking garage, determining the CO evacuation times and its residual concentration under different scenarios.

In this context, the present study is based on the application of CFD simulations to model the behavior of air flow and CO dispersion in an underground multi-family parking garage, comparing different ventilation configurations to identify the most efficient one. For this purpose, the guidelines of the Peruvian regulation RNE-EM030-2020 [12] and the ASHRAE standards [13] are followed, guaranteeing that the proposed solutions are feasible and comply with the air quality requirements.

The general objective is to optimize the ventilation system in multi-family car parks using CFD simulation to ensure effective reduction of carbon monoxide. This work is divided into the following sections. Section II presents the methodology, describing the CFD-based approach to evaluate the effectiveness of the ventilation system and detailing the design and simulation criteria. Section III presents the results and discussion, presenting the results of the simulations, comparing different ventilation configurations and validating the proposed design. Finally, conclusions are drawn on the effectiveness of the designed system and recommendations are made for future implementations.

II. Methodology

A. Materials

The study is based on the modeling of an underground parking garage of a multi-family residential building. The following characteristics are considered for the design of the ventilation system, as shown in Table 1.

Table 1. General parking characteristics.

Parameter	**Basement 1**	**Basement 2**
Parking area	495 m²	495 m²
Level height	2.80 m	2.55 m
Number of parking spaces	11	13
Vehicles in transit	6	7

1.1 Multi-Family Parking Structure Model

The model represents an architectural design with the distribution of parking spaces, vehicle access, and circulation areas. Figure 1 shows the location of carbon monoxide (CO) emission points and fresh air injection points, which are key elements of the ventilation system.

The CO emission points are marked with circles containing yellow rectangles, which indicate the areas where concentrations of polluting gases are generated due to the transit and permanence of vehicles. On the other hand, the air injection points are marked with blue elements that represent the ventilation systems that introduce fresh air to dilute and eliminate the CO accumulated in the environment.

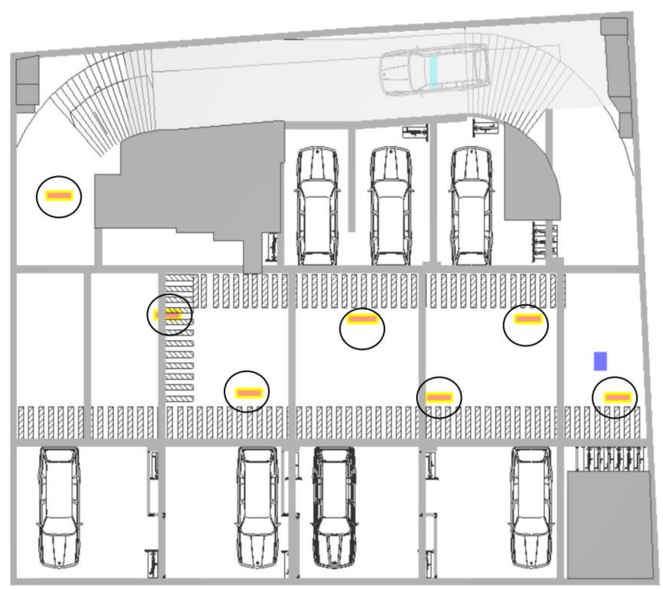

Figure 1. Multi-family parking lot distribution.

The model shows an orderly distribution of parking spaces, circulation areas and access ramps, which makes it possible to analyze the efficiency of the ventilation system in terms of air renewal and control of pollutants in the basement.

1.2 Ventilation Equipment Used

Table 2. Specifications of the Extractors, Injectors and Centrifugal Fans in the Parking Garage Ventilation System.

Level/Location	**Fan Type**	**Flow Rate (CFM)**

Basement 1	Axial extractor	4,075
Basement 1	Axial injector	4,075
Basement 2	Axial extractor	3,741
Basement 2	Axial injector	3,741
Basement 1	Centrifugal Jet Fan (hood type)	1,485
Basement 2	Centrifugal Jet Fan (hood type)	1,485

The ventilation system in the underground garage consists of axial flow extractors and injectors and Jet Fan centrifugal bell-type fans distributed in basements 1 and 2 to optimize air circulation and carbon monoxide (CO) removal, as shown in Table 2.

1.3 Regulations applied

The study applies the standards RNE-EM030-2020 [5] and ASHRAE 62.1 [13] to design an efficient ventilation system in underground car parks that ensures air quality, as shown in Table 3. Three methods are used to calculate the ventilation flow rate: air changes per hour, flow rate per unit area, and maximum allowable CO concentration.

Table 3. Regulations applied in the study.

Regulation	Description
National Building Regulations (RNE-EM030-2020)	Defines the criteria for mechanical ventilation in underground parking facilities in Peru.
ASHRAE Standard 62.1	Establishes minimum ventilation requirements in enclosed spaces, ensuring indoor air quality.
Ventilation Flow Rate Calculation Methods	Three methods are considered: (1) air changes per hour, (2) per unit area, (3) maximum permissible CO concentration.

2. Methods and tools

The mathematical modelling of airflow and CO dispersion is based on the Navier-Stokes equations, which describe the behaviour of fluids in a confined environment. The mass conservation equation, also known as the continuity equation, states that mass in a closed system is conserved over time. It is expressed mathematically in Equation 1 [14].

$$\frac{\partial \rho}{\partial t} + \nabla.(\rho\, \vec{v}) = 0 \tag{1}$$

Where:

ρ is the density of the fluid (kg/m³), t is time (s) and $\vec{v}$ is the fluid velocity vector (m/s). ∇ represents the divergence, which describes the change in density in a control volume.

The Navier-Stokes equations describe how the motion of a fluid changes as a function of time and space. They are applied to fluid dynamics problems such as the analysis of air flow and the dispersion of pollutants in an enclosed car park. Each equation represents the conservation of momentum in one of the three spatial directions (x, y, z).

The Navier-Stokes equations in vector form are presented in [14].

$$\begin{aligned} \frac{\partial(\rho u)}{\partial t} + \nabla.(\rho\, \vec{v}) &= -\frac{\partial p}{\partial x} + u\nabla^2 u \\ \frac{\partial(\rho v)}{\partial t} + \nabla.(\rho\, \vec{v}) &= -\frac{\partial p}{\partial y} + u\nabla^2 v \\ \frac{\partial(\rho w)}{\partial t} + \nabla.(\rho\, \vec{v}) &= -\frac{\partial p}{\partial z} + u\nabla^2 w \end{aligned} \tag{2}$$

Where:

ρ is the density of the air; u, v and w are the components of the velocity in the x, y and z directions respectively; p is the pressure in the fluid and μ is the viscosity of the air. Similarly, $\nabla^2 u$, $\nabla^2 v$ and $\nabla^2 w$ represent the viscous diffusion term of the fluid.

These equations state that the change in the motion of a fluid is influenced by the change in the amount of motion over time, $\partial(\rho u)/\partial t$, which describes the acceleration of the fluid. The effect of pressure ($\partial p/\partial x$), which generates forces that propel the fluid. The viscosity of the air ($\mu\nabla^2 u$), which acts as a resistance to the movement of the fluid, reducing turbulence and dissipating energy [15].

In the same way, they allow us to simulate how air moves in an underground car park and how it affects the removal of CO. CFD simulation can be used to analyze different ventilation schemes and to optimize the design of extractors and injectors. Airflow patterns can be evaluated and areas where CO accumulates can be found to redesign ventilation and improve air quality.

III. Results And Discussion

3.1 Analysis of the CO concentration in the parking lot

The CFD simulation of the multi-family parking garage allowed the CO concentration to be analyzed as a function of time and ventilation system configuration. The initial CO concentration was set at 50 ppm, and the monitoring sensors were placed 1.5 meters above the floor. At minute 0, the CO concentration in the basement is homogeneous and reaches an initial value of 50 ppm throughout the room with no air movement. During the first few minutes, there is no significant reduction in CO levels due to the lack of airflow, as shown in Figure 2.

After two minutes, an initial displacement of air is observed, although CO concentrations remain close to 50 ppm in most of the parking lot. This behavior indicates that the ventilation system is beginning to produce a dilution effect, although the gas concentration does not decrease significantly. At minute 4, ventilation begins to have a more pronounced effect, with CO concentrations gradually decreasing to a range of 35-45 PPM, especially in the areas near the air injectors. As the process continues, values between 35 and 40 PPM are recorded at minute 6, indicating a greater reduction in areas with greater air circulation, indicating that the ventilation system is working effectively.

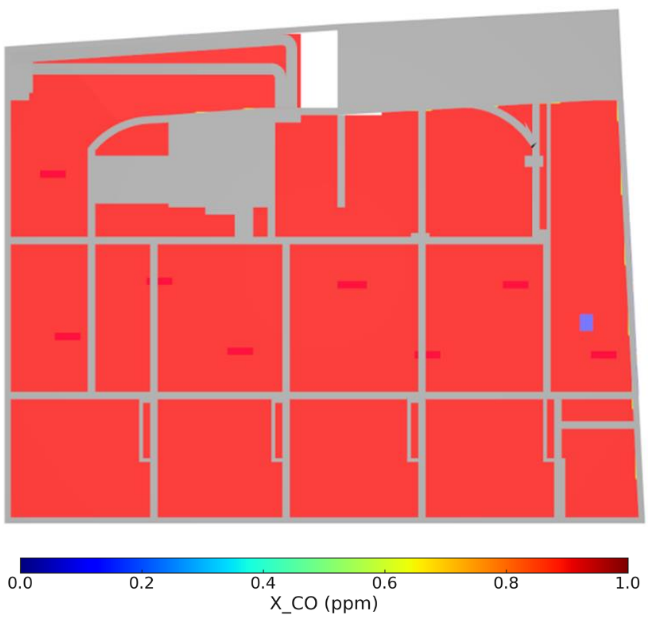

Figure 2. Results of the CFD simulation for CO concentrations at 1.5 m above the ground; t = 0 minutes. Startin the simulation

At minute 8, the central area of Basement 2 has levels below 20 ppm, although some areas still have concentrations around 35 ppm, suggesting the existence of sectors with lower air renewal efficiency. Subsequently, at minute 10, most of the basement registers concentrations below 35 ppm, except for small areas where airflow is less efficient. At minute 12, the basement is almost completely evacuated, and CO levels are very low in most of the space. Finally, at minute 14:30, the system achieves complete evacuation of CO, reaching levels below 35 ppm throughout the garage. These results confirm the effectiveness of the designed ventilation system in ensuring compliance with established air quality standards, as shown in Figure 3.

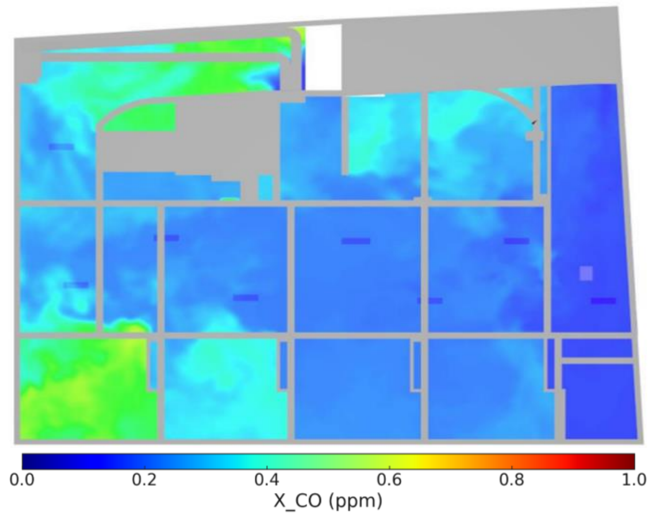

Figure 3. Results of the CFD simulation for CO concentrations at 1.5 m above ground; t = 15 min.

3.2 Comparison of Ventilation Configurations

Two main configurations were compared: conventional ventilation and CFD optimized ventilation.

In terms of CO evacuation time, conventional ventilation showed areas where the CO concentration remained above 35 ppm for more than 13 minutes, which violates ASHRAE 62.1 standards. In contrast, the optimized ventilation reduced the CO evacuation time to 7 minutes, which improved air distribution and accelerated gas dissipation.

Table 4. Evacuation time results in conventional and optimized systems.

Configuration	Evacuation Time (s)	Complies with ASHRAE 62.1?
Conventional Ventilation	800 s (13:20 min)	No
Optimized Ventilation	420 s (7:00 min)	Yes

3.3 Air Velocities in the Basement

Analysis of air velocities in Basement 2 showed significant differences between the two configurations. With conventional ventilation, the average air velocity was 0.1 m/s, which resulted in a longer evacuation time and lower CO dispersion efficiency. In contrast, with optimized ventilation, the air velocity increased to 1.5 m/s, which facilitated rapid dispersion of CO and improved air quality in the evaluated space, as shown in Table 5.

Table 5. Average air velocity for each ventilation configuration.

Configuration	Average Air Velocity (m/s)
Conventional Ventilation	0.1 m/s
Optimized Ventilation	1.5 m/s

3.4. Energy evaluation of the basement ventilation system.

The analysis of the energy consumption of the fans in basement 2 showed significant differences between the two configurations, as shown in Table 6. In the conventional ventilation, the consumption reached 5.2 kWh, while in the optimized ventilation it was reduced to 3.1 kWh, which is a 40% reduction. This reduction was possible because the fans were running for less time without affecting the air quality. In addition, the optimization of the system made it possible to reduce the number of active extractors by improving the distribution of the air flow, which increased the efficiency of the ventilation system.

Table 6. Energy consumption for each ventilation configuration.

Configuration	Energy Consumption (kWh)
Conventional Ventilation	5.2 kWh
Optimized Ventilation	3.1 kWh

The results presented are related to advances in literature. For example, in [16] a decoupled fixed bed gasifier with an integrated syngas burner was designed, simulated and experimentally validated, which allowed optimization of energy efficiency and reduction of emissions. The CFD simulation predicted the temperature and composition of the syngas with high accuracy, with an error of less than 9%. Similarly, in [17], multi-fluid CFD simulation with the Eulerian-Eulerian model was used to analyze the gasification of sawdust in an industrial-scale fluidized bed gasifier and to evaluate the hydrodynamic behavior and conversion of the biomass. Eight heterogeneous and five homogeneous reactions, including the oxidation of CO, CH_4 and H_2, were included to accurately predict the composition of the synthesis gas. The methodology was validated with experimental data, providing a reliable model to optimize the thermal and kinetic performance of the process.

IV. Conclusions

The objective of this work was to optimize the ventilation system of multi-family parking lots using CFD simulation to ensure effective reduction of carbon monoxide. The results show that. The optimized ventilation system reduced the CO evacuation time from 13:20 min to 7:00 min, meeting the requirements of ASHRAE 62.1. Air velocity increased from 0.1 m/s to 1.5 m/s with optimization, improving the efficiency of the ventilation system. Energy consumption was reduced by 40%, lowering operating costs without compromising air quality. Therefore, optimization of the ventilation system using CFD simulation improved CO removal, reduced energy consumption, and ensured compliance with ASHRAE 62.1. These results can be applied to the design and improvement of underground parking garage ventilation systems.

Acknowledgment

We are grateful to the Universidad Tecnológica del Peru (UTP) for supporting part of this research. We also thank CONCYTEC Peru for providing access to various research databases.

References

[1] S. Zhang, Y. Song, W. Deng, and M. Luo, “Planning vs Market-led? Identifying urban-suburbs transition zones in metropolitan expansion: A multi-source data fusion framework,” Information Fusion, vol. 115, p. 102788, Mar. 2025, doi: 10.1016/J.INFFUS.2024.102788.

[2] G. Zhou, H. Zhen, Y. Zhang, G. Li, X. Lv, and X. Zhang, “Synthesis and carbon monoxide purification performance of ZSM-5 molecular sieve Co-doped Mn/V catalytic material,” Sep Purif Technol, vol. 359, p. 130327, Jun. 2025, doi: 10.1016/J.SEPPUR.2024.130327.

[3] F. Orina et al., “Household carbon monoxide (CO) concentrations in a large African city: An unquantified public health burden?,” Environmental Pollution, vol. 351, p. 124054, Jun. 2024, doi: 10.1016/J.ENVPOL.2024.124054.

[4] G. Razzano, S. Brandi, M. S. Piscitelli, and A. Capozzoli, “Rule extraction from deep reinforcement learning controller and comparative analysis with ASHRAE control sequences for the optimal management of Heating, Ventilation, and Air Conditioning (HVAC) systems in multizone buildings,” Appl Energy, vol. 381, p. 125046, Mar. 2025, doi: 10.1016/J.APENERGY.2024.125046.

[5] IPDU, “Normas que modifican RNE” Accessed: Feb. 14, 2025. [Online]. Available: https://ipdu.pe/2024/03/17/normas-que-modifican-rne/

[6] H. bouakline, I. Ziani, M. I. Yahyaoui, F. Corrias, A. Abdeslam, and A. El Bachiri, “Using green solvents as co-solvent in supercritical carbon dioxide extraction: A promising source of extracts rich in antioxidants and antimicrobials agents,” Sustain Chem Pharm, vol. 44, p. 101950, Apr. 2025, doi: 10.1016/J.SCP.2025.101950.

[7] J. Brodny and M. Tutak, “Applying computational fluid dynamics in research on ventilation safety during underground hard coal mining: A systematic literature review,” Process Safety and Environmental Protection, vol. 151, pp. 373–400, Jul. 2021, doi: 10.1016/J.PSEP.2021.05.029.

[8] S. Jo, G. Kim, and M. Sung, “A study on contaminant leakage from Airborne Infection Isolation room during medical staff entry; Implementation of walking motion on hypothetical human model in CFD simulation,” Journal of Building Engineering, vol. 86, p. 108812, Jun. 2024, doi: 10.1016/J.JOBE.2024.108812.

[9] S. Diao, H. Li, J. Wang, C. Wei, Y. Yao, and M. Yu, “Hydrogen leakage location prediction for fuel cell vehicles in parking lots: A combined study of CFD simulation and CNN-BiLSTM modeling,” Int J Hydrogen Energy, vol. 109, pp. 115–128, Mar. 2025, doi: 10.1016/J.IJHYDENE.2025.02.065.

[10] H. Sun et al., “Pollutant cross-transmission in courtyard buildings: Wind tunnel experiments and computational fluid dynamics (CFD) evaluation,” Build Environ, vol. 264, p. 111919, Oct. 2024, doi: 10.1016/J.BUILDENV.2024.111919.

[11] J. Cho, J. Kim, W. Park, and Y. Kim, “A step-by-step numerical method for optimization of mechanical ventilation in deep underground enclosed parking lots: A case-design study,” Journal of Building Engineering, vol. 42, p. 102799, Oct. 2021, doi: 10.1016/J.JOBE.2021.102799.

[12] C. y S. Ministerio de Vivienda, “Modificación de la Norma Técnica EM.030 Instalaciones de Ventilación del Reglamento Nacional de Edificaciones.” Accessed: Feb. 14, 2025. [Online]. Available: https://www.gob.pe/institucion/vivienda/normas-legales/1210577-232-2020-vivienda

[13] ANSI/ASHRAE Standards, "Standards 62.1 & 62.2." Accessed: Feb. 14, 2025. [Online]. Available: https://www.ashrae.org/technical-resources/bookstore/standards-62-1-62-2?utm_source=chatgpt.com
[14] Z. Guo and W. Wang, "On the uniqueness and non-uniqueness of the steady planar Navier-Stokes equations in an exterior domain," J Differ Equ, vol. 428, pp. 483–510, May 2025, doi: 10.1016/J.JDE.2025.02.031.
[15] N. Nouaime, B. Després, M. A. Puscas, and C. Fiorini, "Sensitivity analysis for incompressible Navier–Stokes equations with uncertain viscosity using polynomial chaos method," European Journal of Mechanics - B/Fluids, vol. 111, pp. 308–318, May 2025, doi: 10.1016/J.EUROMECHFLU.2025.01.012.
[16] C. Siripaiboon, P. Sitthichirachat, P. Sarabhorn, K. Somkeattikul, C. Areeprasert, and F. Scala, "Design, CFD simulation, and experimental verification of a novel fixed-bed decoupling gasifier with an integrated syngas burner using high-temperature flue gas," Process Safety and Environmental Protection, p. 106883, Feb. 2025, doi: 10.1016/J.PSEP.2025.106883.
[17] V. Singh, P. Nemalipuri, V. Vitankar, and H. C. Das, "Multifluid computational fluid dynamics simulation for sawdust gasification inside an industrial scale fluidized bed gasifier," Mater Today Proc, Jun. 2023, doi: 10.1016/J.MATPR.2023.06.258.