Journal of Al-Azhar University Engineering Sector

Vol.20, No. 76, July 2025, 909-919

EXPERIMENTAL AND NUMERICAL INVESTIGATION OF BUILDING SMOKE MANAGEMENT

Mohamed I. Younes[1*], ElAdl A. Elkady[1], Mustafa A. Mohamed[1]

[1]Mechanical Engineering Department, Faculty of Engineering, Al-Azhar University, Nasr City, Cairo, Egypt

* **Correspondence**: m.younes1711@gmail.com

Citation:

M. I. Younes, E. A. Elkady, M. A. Mohamed, " Experimental and Numerical Investigation of Building Smoke Management ", Journal of Al-Azhar University Engineering Sector, vol. 20, pp. 909 - 919, 2025.

Received: 13 February 2025

Revised: 10 May 2025

Accepted: 31 May 2025

Doi: 10.21608/auej.2025.360338.1779

ABSTRACT

Hot smoke from fires poses a major threat to the safety of both building occupants and firefighters, with smoke inhalation responsible for around 85% of fire-related deaths. The extreme temperatures of smoke can cause serious burns to the skin and respiratory system, and extended exposure may lead to fatal injuries. To address this hazard, the present study conducts a thorough experimental and numerical analysis using Computational Fluid Dynamics (CFD) to enhance Smoke Management Systems (SMS) in enclosed spaces. The research specifically examines the role of Air Changes per Hour (ACH) in reducing smoke-related risks. Temperature trends over time were recorded for three different ACH levels. Findings highlight ACH as a critical factor influencing system performance in confined environments. Additionally, A CFD simulation was conducted for one of the test scenarios, and the results exhibited a comparable trend that closely matched the experimental data, supporting the CFD model's potential for broader applications in smoke control planning.

KEYWORDS: Smoke Management Systems, Air Change per Hour, Experimental and CFD.

دراسة تجريبية وعددية لادارة الدخان في المباني

محمد ابراهيم يونس[١*]، العدل احمد القاضي[١]، مصطفي علي محمد[١]

[١]قسم الهندسة الميكانيكية، كلية الهندسة، جامعة الأزهر، مدينة نصر، ١١٨٨٤، القاهرة، مصر

البريد الالكتروني للباحث الرئيسي: m.younes1711@gmail.com

الملخص

الدخان الساخن الناتج عن الحرائق يشكل تهديدًا كبيرًا لسلامة كل من الأشخاص ورجال الإطفاء، حيث يعد استنشاق الدخان السبب الرئيسي للوفيات المرتبطة بالحرائق، حيث يُسَبب حوالي ٨٥٪ من حالات الوفاة. يمكن للحرارة العالية للدخان أن تسبب حروقًا شديدة للجلد و مشاكل للجهاز التنفسي، ومع التعرض المطول قد تؤدي إلى إصابات تهدد الحياة أو الوفاة. استجابةً لهذه المشكلة، يقوم هذا البحث بإجراء دراسة تجريبية ومحاكاه لتحسين أنظمة إدارة الدخان في المساحات المغلقة. يركز البحث على تأثير معدل تغييرات الهواء في الساعة (ACH) ويهدف إلى تقييم أهميته في التخفيف من المخاطر المرتبطة بالدخان. تم تسجيل التغيرات في درجة الحرارة مع مرور الوقت لثلاثة معدلات مختلفة من (ACH) تؤكد النتائج على التأثير الكبير لـ معدل تغييرات الهواء في الساعة على أداء الأنظمة المغلقة. تم أيضًا إجراء محاكاة باستخدام برنامج محاكاة لأحد التجارب، وأظهرت المقارنة بين النتائج التجريبية والمحاكاه توافقًا، مما يشير إلى أن نموذج CFD هو أداة قابلة للتطبيق في التطبيقات واسعة النطاق ويمكن أن تساهم في تحسين استراتيجيات التحكم في الدخان.

الكلمات المفتاحية : ادارة الدخان ، معدل تغير الهواء، نموزج محاكاه، تجارب عملية.

1. INTRODUCTION

Smoke generated from building fires poses a significant threat to both occupant and firefighter safety. Smoke inhalation is the leading cause of fire-related fatalities, accounting for approximately 85% of deaths in such incidents [1]. Hot smoke is particularly hazardous due to its intense thermal energy, which can cause severe burns to the skin and respiratory tract. In extreme cases, prolonged exposure may result in life-threatening injuries or death [2]. Additionally, the combined effects of heat, smoke, and flames can compromise the structural integrity of buildings by weakening steel reinforcements within concrete, potentially leading to structural collapse [3].

Enclosed spaces such as underground car parks, shopping malls, and building corridors require stringent control of indoor air quality, primarily due to the accumulation of toxic gases like carbon monoxide. Maintaining adequate ventilation—whether through natural systems or mechanical solutions such as ductless jet fan systems—is essential for ensuring air quality and effective smoke management [4]. Among the critical parameters influencing smoke control systems is the Air Change per Hour (ACH) rate, which directly affects the removal and dispersion of smoke during fire events [5].

A more sophisticated approach to design is essential for underground car parks, not only to enhance accessibility but also to ensure safety by addressing challenges related to indoor air quality and smoke management. Traditional empirical calculations may be prone to errors due to their inability to fully capture the complexities of smoke flow under dynamic conditions. In contrast, Computational Fluid Dynamics (CFD) simulations offer a more accurate and practical method for visualizing fluid behavior and predicting fire smoke dynamics [6, 7]. Validation of CFD models through experimental studies is essential for improving their accuracy and ensuring reliable application in real-world scenarios.

Previous research has explored various aspects of ventilation and smoke control in enclosed environments. A combined experimental and numerical study demonstrated the effectiveness of natural ventilation in removing fire-induced gases in underground car park fire lanes, offering practical guidance for future fire safety designs [8]. Additionally, the development of impulse ventilation systems has marked a significant advancement in car park ventilation. A study conducted in Riyadh, Saudi Arabia, found that a system utilizing 11 jet fans enhanced evacuation efficiency and reduced carbon monoxide levels more effectively than alternative configurations [10]. CFD played a key role in these assessments, enabling precise predictions of air flow patterns, temperature distribution, and smoke density within the space [9, 10].

Further research involving scaled-down subway models revealed that smoke temperature followed an exponential decay pattern, with irregularities observed in transverse temperature distributions. Empirical models were developed to accurately predict both longitudinal and transverse smoke temperatures, improving risk assessments for similar confined structures [11]. Similarly, a study investigating room–corridor configurations established correlations between heat release rate (HRR) and room smoke temperature. In impingement areas, maximum corridor smoke temperature was found to depend solely on HRR, whereas in non-impingement areas, it also varied with distance. The study proposed dimensionless correlations to aid fire modeling in such configurations [12].

In the context of evacuation safety, a study utilizing PyroSim and Pathfinder simulations assessed fire scenarios in a university teaching building. The analysis highlighted the influence of smoke, temperature, and visibility on occupant evacuation, with critical conditions arising just 105 seconds after ignition. Overcrowding—particularly beyond 150–180 occupants per floor—significantly hindered safe evacuation, underscoring the need for strategic fire safety planning in educational facilities [20].

In another study, numerical simulations using PyroSim investigated smoke spread and temperature distribution in a large indoor pedestrian street. Results showed that higher heat release rates led to faster smoke propagation and elevated temperatures, reaching up to 400 °C. Smoke exhaust systems were found to reduce both temperature and smoke layer thickness, effectively shortening the stable fire phase. Mechanical exhaust systems outperformed natural ventilation, achieving greater attenuation of smoke with coefficients varying by 22% and 13%, respectively [21].

As such, this study aims to comprehensively examine the impact of Air Changes per Hour (ACH) on smoke temperature by integrating scaled experimental analysis with advanced Computational Fluid Dynamics (CFD) simulations. The objective is to gain deeper insights into airflow dynamics and thermal behavior in smoke-filled environments, while also validating the reliability of CFD models for large-scale applications to optimize time and resources in smoke management.

2. Materials and Methods

2.1. Mathematical modeling

The mathematical basis for simulating smoke management requires solving key equations related to fluid dynamics, heat transfer, and smoke dispersion. Computational Fluid Dynamics (CFD) simulations are an essential tool for analyzing systems that involve mass and heat transfer, providing a more detailed and advanced understanding of these phenomena through computer models. The Fire Dynamics Simulator (FDS) has gained popularity as a CFD tool for describing fire evolution [6, 13, 14] employing a large eddy simulation form of the Navier–Stokes equations tailored for low-speed, thermally-driven flow [6]. PyroSim, a graphical user interface for the Fire Dynamics Simulator (FDS), facilitates this mathematical modeling process [15-17].

Governing Equations:

The core of the mathematical model is formed by the continuity, momentum equations (Navier-Stokes equations), which represent mass conservation and fluid motion, respectively [18]. The unsteady 3D continuity equation can be represented as:

$$\frac{\partial \rho}{\partial t} + \frac{\partial(\rho u)}{\partial x} + \frac{\partial(\rho v)}{\partial y} + \frac{\partial(\rho w)}{\partial z} = 0 \qquad (1)$$

While the 3D momentum equations are presented as fellow:

$$\frac{\partial(\rho u)}{\partial t} + \frac{\partial(\rho u^2)}{\partial x} + \frac{\partial(\rho uv)}{\partial y} + \frac{\partial(\rho uw)}{\partial z} = -\frac{\partial p}{\partial x} + \mu\left(\frac{\partial^2 u^2}{\partial x^2} + \frac{\partial^2 u^2}{\partial y^2} + \frac{\partial^2 u^2}{\partial z^2}\right) + \rho g_x \qquad (2)$$

$$\frac{\partial(\rho v)}{\partial t} + \frac{\partial(\rho vu)}{\partial x} + \frac{\partial(\rho v^2)}{\partial y} + \frac{\partial(\rho vw)}{\partial z} = -\frac{\partial p}{\partial y} + \mu\left(\frac{\partial^2 v^2}{\partial x^2} + \frac{\partial^2 v^2}{\partial y^2} + \frac{\partial^2 v^2}{\partial z^2}\right) + \rho g_y \qquad (3)$$

$$\frac{\partial(\rho w)}{\partial t} + \frac{\partial(\rho wu)}{\partial x} + \frac{\partial(\rho wv)}{\partial y} + \frac{\partial(\rho w^2)}{\partial z} = -\frac{\partial p}{\partial z} + \mu\left(\frac{\partial^2 w^2}{\partial x^2} + \frac{\partial^2 w^2}{\partial y^2} + \frac{\partial^2 w^2}{\partial z^2}\right) + \rho g_z \qquad (4)$$

Where, ρ represents density, u, v, w are velocity components in the *x, y, z* directions, p is pressure, μ is dynamic viscosity, and g_x, g_y, g_z are gravitational accelerations.

Energy conservation equation

$$\rho c_p\left(\frac{\partial T}{\partial t} + u\cdot\nabla T\right) - \frac{dp_0}{dt} = \dot{q} + \nabla(k\nabla T)$$

where T is the temperature, p_0 is the environmental pressure. $\dot{q}$ is the specified volumetric heat source, cp is the specific heat at constant pressure, and k is the heat conductivity coefficient.

2.2. Experimental set-up

A series of experiments were carried out using a scaled room model. The scaled model is 1.5 meters in length, 1 meter in width, and 1 meter in height, with one transparent wall, as seen in Fig. 1. This experimental setup was equipped with an axial inline exhaust fan and a fresh air fan, both of which were connected to the room through a duct system. In Fig. 1, item B and D illustrate the exhaust fan and a control volume damper, each with dimensions of 10 cm by 10 cm, which are connected to an exhaust wall opening. These dampers play a crucial role in regulating the airflow and can be adjusted to control the airflow rate, which in turn influences the Air Changes per Hour (ACH) within the room.

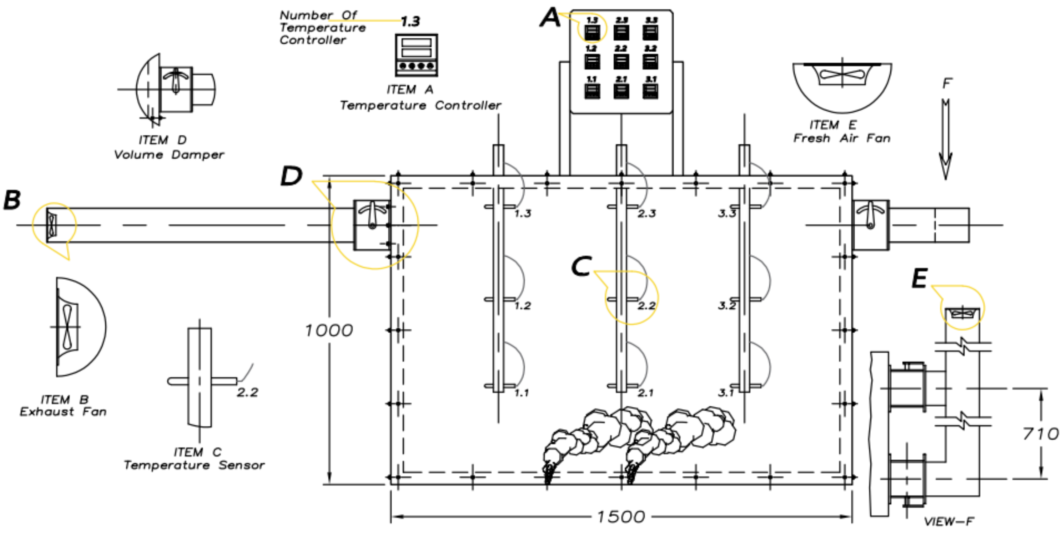

Fig. 1: Schematic diagram for the experimental setup

Items E and F in **Fig. 1** show the fresh air fan and two additional control volume dampers, each also measuring 10 cm by 10 cm, connected to openings in the room. These dampers are used to regulate the intake of fresh air, allowing for precise control over the ventilation and ensuring the air quality is maintained at the desired levels. The control volume dampers in both the exhaust and fresh air systems enable precise adjustments to the ventilation setup, simulating various air exchange and smoke management conditions, with the assistance of an air anemometer that measures duct velocity.

Additionally, three groups of Type K thermocouples (Groups 1, 2, and 3), as shown in item C of **Fig. 1**, were arranged horizontally across the room. Each group consisted of three thermocouples positioned vertically, labeled as 1.1, 1.2, 1.3, and so on, to measure the temperature at various locations throughout the room. These thermocouples are essential for capturing temperature variations in the room, which are critical for understanding the airflow and heat distribution during the experiments. Each thermocouple was connected to a temperature controller, as indicated in item A of the figure, which provided real-time temperature readings and allowed precise monitoring of temperature changes within the room during the experiment. **Table 1** provides an overview of the technical specifications of the instruments used for the measurements.

Table.1: Technical Specifications of the Measuring Instruments

Measured parameters	Device Name	Range	Accuracy
Temperature	Thermocouples Type K	From (-270 to 1260C)	+/- 2.2C or +/- 0.75%
Air velocity	Hot wire anemometer	0.2 to 40 m/sec	±3% of reading
Temperature Controller	Digital Controller-REX	as per Thermocouples range	±0.5% of display value + 1 digit)

All thermocouples have been calibrated. Fig. 2 illustrate sample of thermocouples calibration curves. Calibration is made by using iced and boiling water.

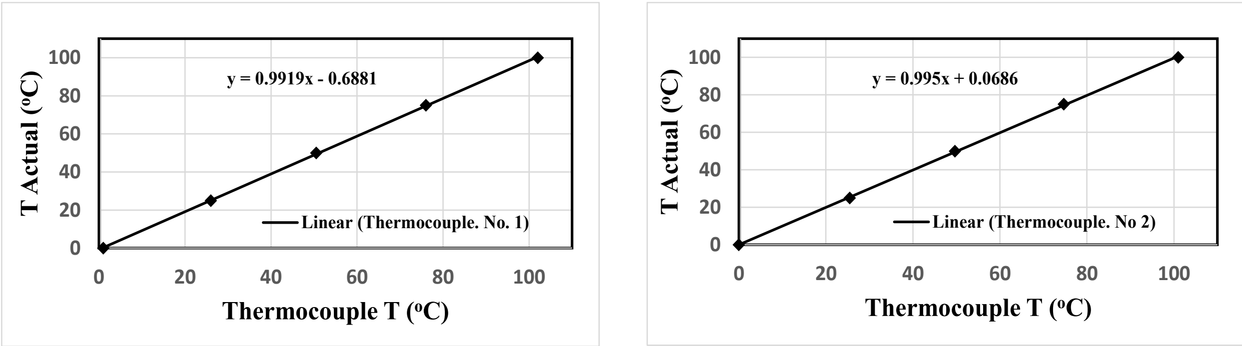

Fig. 2: Sample of Thermocouples Celebration Curves.

This setup was constructed at Al-Azhar University, and **Fig. 3** presents the actual view of the experimental arrangement, highlighting the key components and their physical configuration within the laboratory environment. The overall experimental design and setup were carefully planned to simulate various ventilation scenarios and study the effects of different Air Changes per Hour (ACH) on the room's temperature and air quality, thereby providing valuable insights into the dynamics of smoke management and ventilation systems. In this study, three Air Changes per Hour (ACH) rates 8, 10, and 12 were tested to evaluate their impact on room ventilation, smoke temperature, and air quality. These varying ACH levels were chosen to analyze the effectiveness of different ventilation rates in managing indoor air conditions.

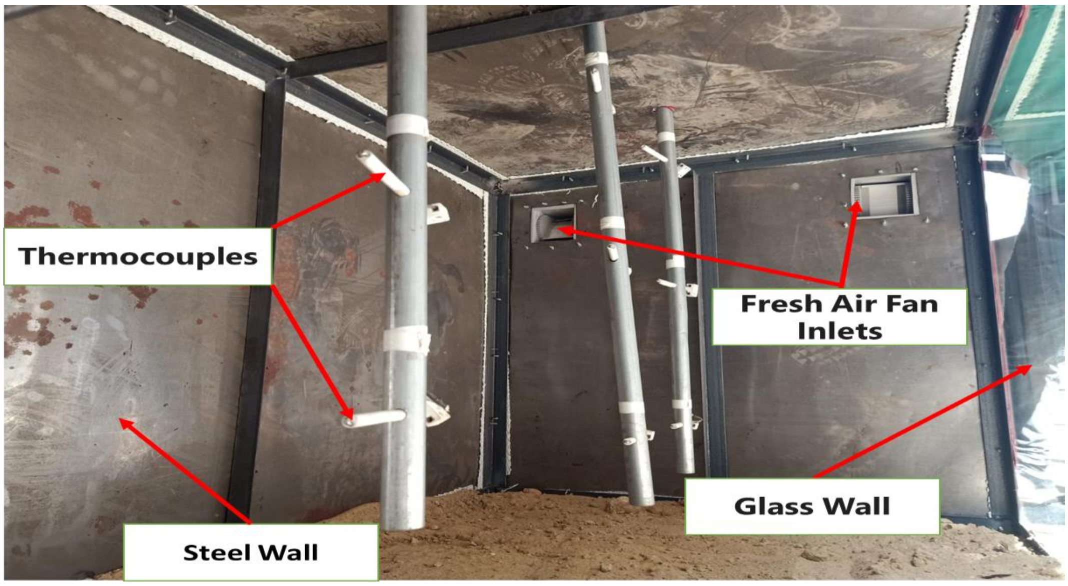

Fig. 3: Actual view of the experimental set-up

2.3. FDS Model description

A Computational Fluid Dynamics (CFD) model, replicating the exact dimensions of the experimental setup, was developed using the fire dynamics simulation software PyroSim. The default configuration of the Fire Dynamics Simulator (FDS) includes key features such as low-Mach-number Large Eddy Simulation (LES), second-order, kinetic-energy-conserving numerical schemes, and a structured, uniform, staggered grid. The model also incorporates a simple immersed boundary method for representing flow obstructions, a lumped-species approach for simplified combustion chemistry, and a Deardorff-based sub grid-scale eddy viscosity model. Additional features include constant turbulent Schmidt and Prandtl numbers, the eddy dissipation concept for single-step fuel–oxidizer reactions, and gray gas radiation modeling via the finite volume method for solving the radiation transport equation [17].

Fig. 4 presents a simplified representation of the model, illustrating its geometric configuration for computational analysis. An air change rate of 10 ACH was simulated, with temperature thermocouples placed at the same locations as in the experimental setup to ensure consistency in data comparison.

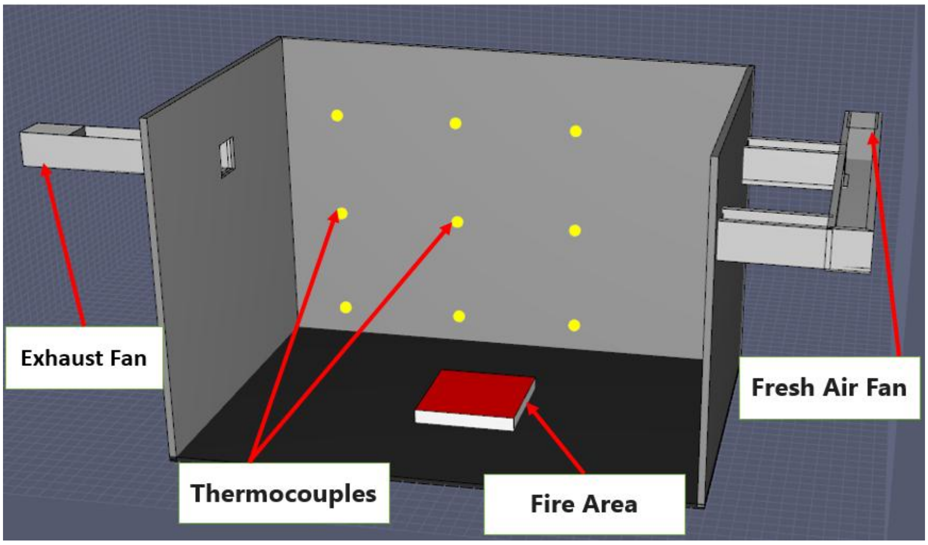

Fig. 4: Simplified geometry of the model used for computational analysis.

The mesh was designed to cover the entire model, as the choice of mesh size significantly influences the accuracy of the simulation results. A smaller mesh size reduces numerical fluctuations, improving the precision of the results, but also increases the computational time. Consequently, selecting an optimal mesh size is crucial for achieving a balance between simulation accuracy and computational efficiency in fire simulations.

To analyze the gride sensitivity the dimensionless expression D* is given in the FDS Operation Manual[16], and δx is the nominal size of the grid cell. Its definition formula is as follows:

$$D^* = \left[\frac{Q}{\rho_0 c_p T_0 \sqrt{g}}\right]^{2/5}$$

where D^* is the characteristic diameter of fire, m; Q is the heat release rate, kW; g is the acceleration of gravity, m/s^2; ρ_0 is the ambient air density, kg/m^3; c_p is the specific heat capacity at constant pressure, kJ/(kg·K); T_0 is the ambient air temperature, K, the ratio of (D*/δx) shall be from 4-16 19,20].

Due to the computational limitations associated with using a personal computer, the (D*/δx) ratio was constrained to approximately ٨. A uniform grid size of 0.0٢٨ meters was employed across all fire scenarios presented in this study to maintain consistency and ensure manageable simulation times. This resolution was selected as a compromise between numerical accuracy and computational feasibility. As a result, the model consisted of a total of ٣١٥,٠٠٠ cubic cells, adequately capturing the essential features of the physical domain while allowing for efficient processing within the available hardware constraints. The corresponding mesh structure is illustrated in **Fig. 5**.

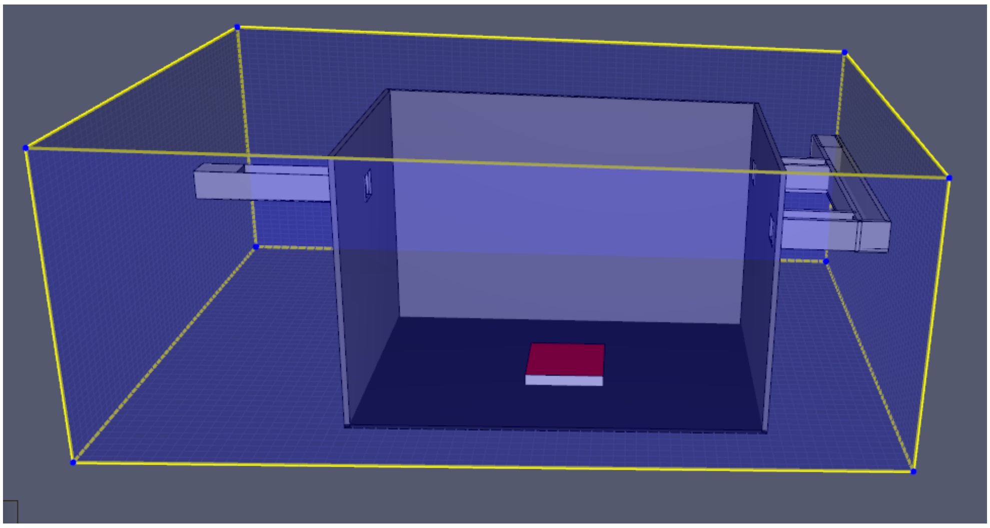

Fig. 5: Mesh structure of the model with a uniform cell size of 0.04 m.

3. RESULTS AND DISCUSSION

3.1. Experimental results

To investigate the impact of Air Changes per Hour (ACH) on room temperature, a series of controlled experiments were carried out for a duration of 750 seconds at each specified ACH rate. During these experiments, temperature readings were systematically recorded every 20 seconds from all strategically placed thermocouples within the experimental setup. These measurements provided detailed data on the temperature variations within the room, allowing for a comprehensive analysis of the thermal behavior under different ventilation conditions.

Fig. 6 presents a comparative analysis of the temperature profiles from all thermocouples, highlighting the trends observed across the different ACH rates. At each ACH level, the temperature exhibited a rapid initial increase until it reached a maximum value. After attaining this peak temperature, the temperature began to decrease gradually, eventually reaching a point of stabilization. This pattern was consistently observed across all ACH rates, indicating a predictable thermal response to the changes in ventilation.

Group 2 (2.1, 2.2, 2.3) recorded the highest temperatures due to its proximity to the fire source. Group 1 (1.1, 1.2, 1.3) and Group 3 (3.1, 3.2, 3.3) recorded lower temperatures, as Group 1 was located near the exhaust wall opening and Group 3 near the fresh air intake, both at the same level. However, there was a temperature difference between the low-level thermocouples (1.1, 3.1) and the high-level ones (1.3, 3.3) which can be attributed to the buoyant rise of hot smoke. It is evident that temperature decreases as the air change rate increases.

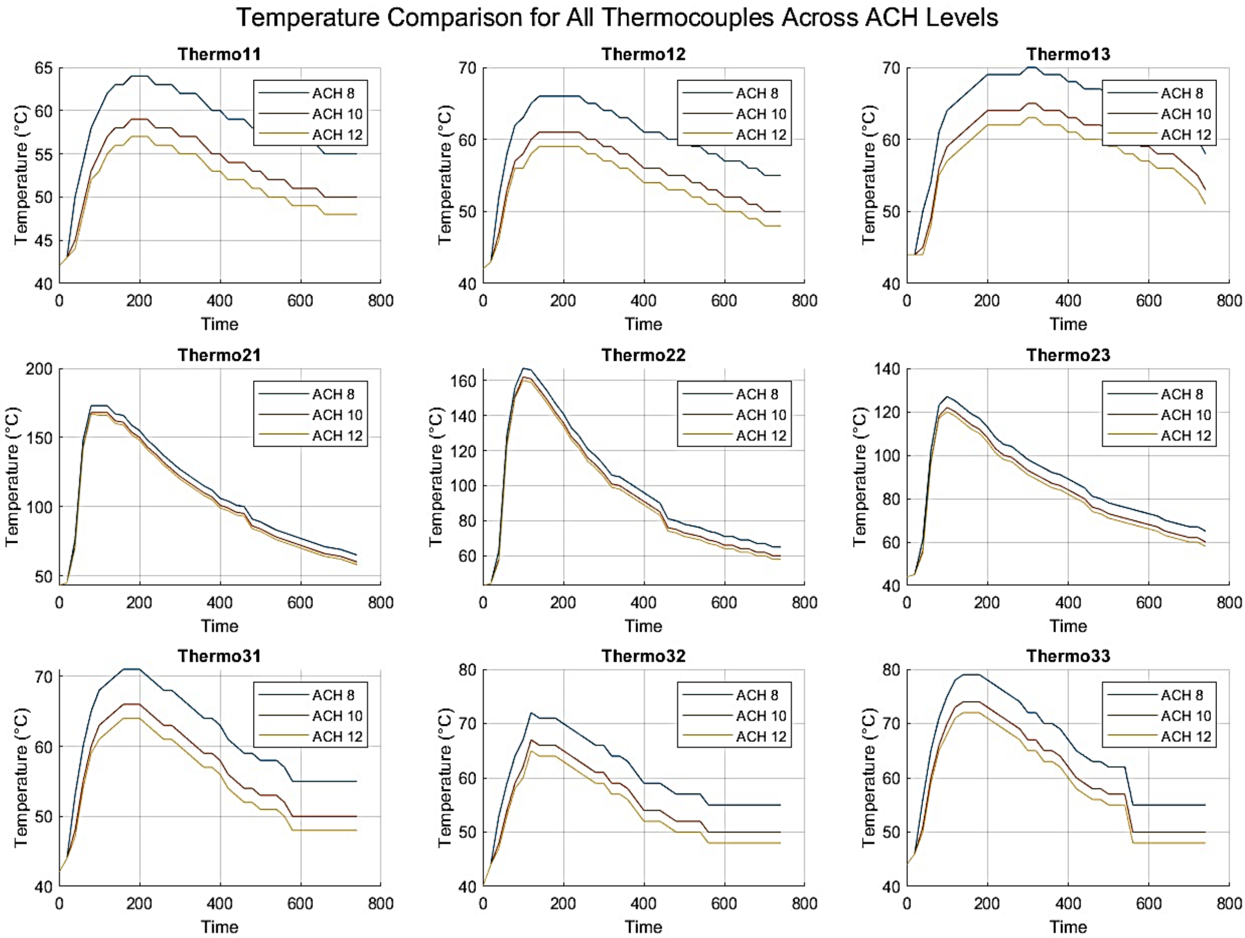

Fig. 6: Experimental results for various Air chang per hours

3.2. Comparison between experimental and CFD simulation results

A CFD model simulating an Air Change per Hour (ACH) of 10 was developed using the fire dynamics simulation tool "Pyrosim," with dimensions matching those of the experimental setup.

Fig. 7 presents contour maps of velocity, temperature, and visibility, which collectively illustrate the spatial distribution and dynamic variations in airflow patterns, thermal conditions, and visual obscuration within the simulated environment during the fire scenario. These visualizations provide critical insight into how the fire influences internal environmental conditions over time. The velocity contours reveal the direction and intensity of air movement, indicating how smoke and heat are transported throughout the space. Notably, areas near the fresh air inlet and exhaust air openings exhibit higher airflow velocities, driven by ventilation forces and pressure differences. Similarly, elevated velocities are also observed near the fire source due to thermal buoyancy effects and the rapid rise of hot gases.

Temperature distributions highlight zones of high thermal intensity, aiding in identifying areas most affected by the fire's heat. Meanwhile, visibility contours depict regions with reduced clarity due to smoke accumulation, which is essential for evaluating evacuation safety and occupant tenability. Together, these contours offer a comprehensive understanding of the fire's impact on the indoor environment, supporting effective fire safety assessment and emergency planning.

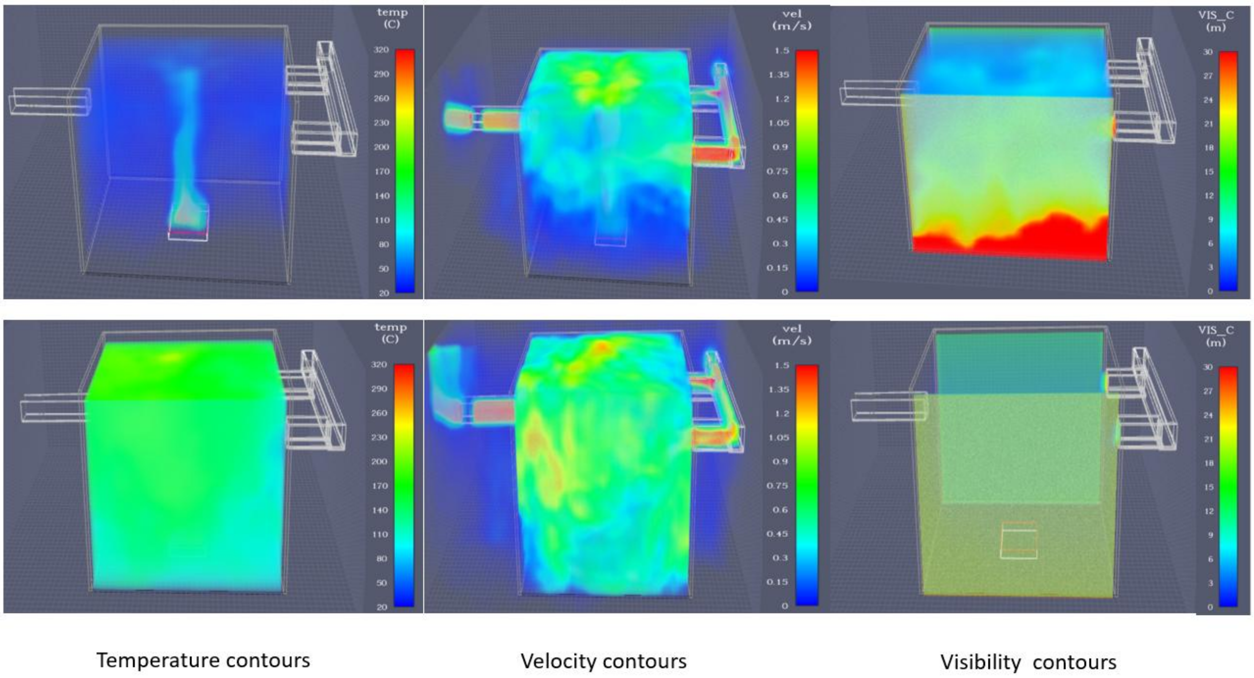

Fig. 7: 3-D Velocity, temperature, and visibility contours showing airflow patterns, heat spread, and smoke accumulation during the fire.

To facilitate a rigorous comparison between the CFD simulation results and the experimental data, temperature recordings from CFD model thermocouples were extracted and compared with those from the experimental setup. This comparison is illustrated in **Fig. 8**.

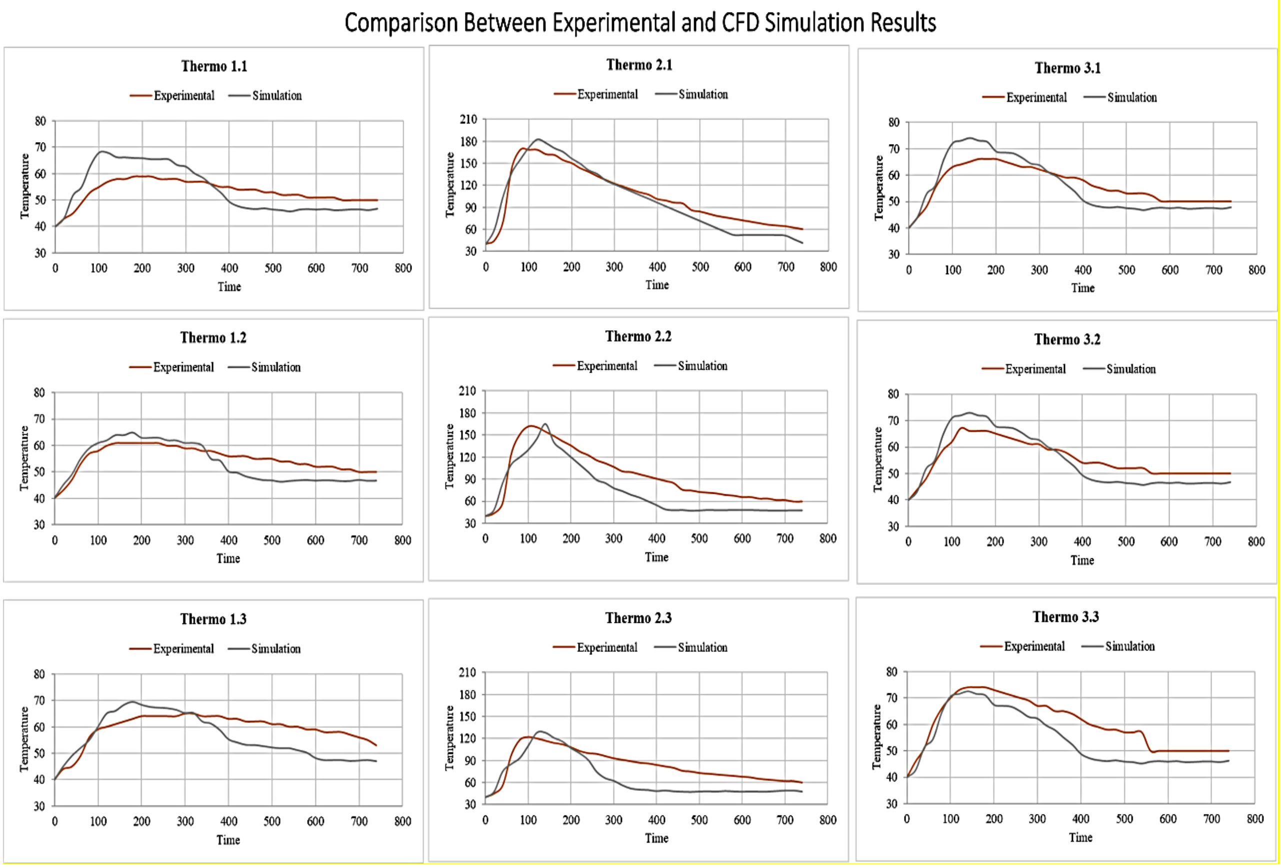

Fig. 8: Comparison between experimental and CFD simulation results

A detailed analysis of the graphs presented reveals that the temperature profiles from both the simulation and experimental results exhibit comparable trends. Specifically, the temperature rises rapidly during the initial stage, reaches a peak, and then gradually declines until stabilizing at a steady state. While the overall behavior of the two datasets remains consistent, some observable differences were identified. These discrepancies are primarily attributed to the idealized assumptions inherent in the Computational Fluid Dynamics (CFD) model. Such simplifications including assumptions like ideal combustion assumption ,time of ignition and HRR vs time are often necessary to model complex physical processes but can result in deviations between simulated and experimental outcomes. These variations are expected due to the intrinsic limitations of the CFD approach and the simplifications employed in the modeling process.

CONCLUSIONS

The effect of Air Changes per Hour (ACH) rates on smoke control strategies with three different values was experimentally studied. The main finding was the higher the ventilation rates the lower the temperature ranges inside the controlled volume. Higher air changes reduced the number of stagnation points. One experiment was used to benchmark CFD approach for further use in digital prototyping smoke control strategies. The experimental and simulation results were in a fair agreement. CFD with Large eddy simulation (LES) approach, could be a reliable tool for predicting the general trends, especially when applied to large-size scenarios. The suggested modeling approach of benchmarking small size models with experimental measurement has the potential to reduce the cost and complexity associated with physical testing for real life enclosed environments.

REFERENCES

[1] Y. Alarie, “Toxicity of Fire Smoke,” Critical Reviews in Toxicology, vol. 32, no. 4. Informa UK Limited, pp. 259–289, Jan. 2002. doi: 10.1080/20024091064246.

[2] Huo, R.; Hu, Y.; Li, Y.Z. Introduction to Building Fire Safety Engineering; China Science and Technology University Press: Hefei, China, pp. 90–91, 2009

[3] J. He, X. Huang, X. Ning, T. Zhou, J. Wang, and R. K. Kit Yuen, “Modelling fire smoke dynamics in a stairwell of high-rise building: Effect of ambient pressure,” Case Studies in Thermal Engineering, vol. 32. Elsevier BV, p. 101907, Apr. 2022. doi: 10.1016/j.csite.2022.101907.

[4] R. Al-Waked, A. Yassin, A. Adwan, and D. B. Mostafa, “Effects of Cross Level Air Interaction within Multilevel Underground Carparks on Indoor Air Quality,” Fluids, vol. 5, no. 4. MDPI AG, p. 177, 2020. doi: 10.3390/fluids5040177

[5] ASHRAE handbook: heating, ventilating, and air-conditioning systems and equipment, Inch-Pound edition. Atlanta, GA: ASHRAE, 2020.

[6] M. Kmecová, P. Buday, J. Vojtaššák, and M. Krajčík, “Design of Fire Ventilation System for an Underground Car Park by CFD Simulations,” AMM, vol. 887, pp. 459–466, 2019, doi: 10.4028/www.scientific.net/AMM.887.459.

[7] W. Binbin, “Comparative Research on FLUENT and FDS’s Numerical Simulation of Smoke Spread in Subway Platform Fire,” Procedia Engineering, vol. 26, pp. 1065–1075, 2011, doi: 10.1016/j.proeng.2011.11.2275.

[8] X. Liu, S. Lu, and Y. Huang, “Experimental Study on Smoke Exhaust of Circular Fire Lane in Underground Vehicle Base of Urban Railway,” IOP Conf. Ser.: Earth Environ. Sci., vol. 455, no. 1, p. 012138, 2020, doi: 10.1088/1755-1315/455/1/012138.

[9] X. Deckers, S. Haga, B. Sette, and B. Merci, “Smoke control in case of fire in a large car park: Full-scale experiments,” Fire Safety Journal, vol. 57. Elsevier BV, pp. 11–21, 2013. doi: 10.1016/j.firesaf.2012.10.017.

[10] J. Glasa, L. Valasek, P. Weisenpacher, and L. Halada, “Cinema Fire Modelling by FDS,” Journal of Physics: Conference Series, vol. 410. IOP Publishing, p. 012013, 2013. doi: 10.1088/1742-6596/410/1/012013.

[11] M. Peng, X. Cheng, K. He, W. Cong, L. Shi, and R. Yuen, “Experimental study on ceiling smoke temperature distributions in near field of pool fires in the subway train,” Journal of Wind Engineering and Industrial Aerodynamics, vol. 199. Elsevier BV, p. 104135, 2020. doi: 10.1016/j.jweia.2020.104135.

[12] Z. Xing, J. Mao, Y. Huang, J. Zhou, W. Mao, and F. Deng, “Scaled Experimental Study on Maximum Smoke Temperature along Corridors Subject to Room Fires,” Sustainability, vol. 7, no. 8. MDPI AG, pp. 11190–11212, 2015. doi: 10.3390/su70811190

[13] R. Rahif and S. Attia, “CFD Assessment of Car Park Ventilation System in Case of Fire Event,” Applied Sciences, vol. 13, no. 18. MDPI AG, p. 10190, 2023. doi: 10.3390/app131810190.

[14] A. Nazari, M. Jafari, N. Rezaei, F. Taghizadeh-Hesary, and F. Taghizadeh-Hesary, “Jet fans in the underground car parking areas and virus transmission,” Physics of Fluids, vol. 33, no. 1. AIP Publishing, 2021. doi: 10.1063/5.0033557.

[15] M. Patil, Design and Analysis of Smoke and Fire in Enclosed Spaces Using CFD, M.S. thesis, VIT University, 2016.

[16] H.-W. Yao et al., “Numerical Simulation of Fire in Underground Commercial Street,” Computational Intelligence and Neuroscience, vol. 2022. Hindawi Limited, pp. 1–9, 2022. doi: 10.1155/2022/4699471.

[17] K. McGrattan, S. Hostikka, R. McDermott, J. Floyd, C. Weinschenk, and K. Overholt, “Fire Dynamics Simulator Technical Reference Guide Volume 1: Mathematical Model,” NIST Special Publication, vol. 1.

[18] Y. Li and R. Xiang, “Particulate pollution in an underground car park in Wuhan, China,” Particuology, vol. 11, no. 1. Elsevier BV, pp. 94–98, 2013. doi: 10.1016/j.partic.2012.06.010.

[19] NIST Special Publication 1019, April 2015. Fire Dynamics Simulation (Version 6.2.0)—User’s Guide; National Institute of Standards and Technology: Gaithersburg, MD, USA, 2015.

[20] M. Jing, G. Zhang, S. Guo, and C. Wang, “Simulation method for fire evacuation safety of teaching buildings in colleges and universities,” Results in Engineering, vol. 25, p. 104512, 2025, doi: 10.1016/j.rineng.2025.104512.

[21] W. Lin et al., “Numerical Simulation on Smoke Temperature Distribution in a Large Indoor Pedestrian Street Fire,” Fire, vol. 6, no. 3, p. 115, 2023, doi: 10.3390/fire6030115.