

CFD Simulation of Smoke Dispersion in Underground Parking Garages

Van Quan Hoang^{1,*} and Quang Dai Do¹

¹Faculty of Engineering and Technology, Thai Nguyen University of Information and Communication Technology, Thai Nguyen 250000, Vietnam

*Corresponding author: hvquan@ictu.edu.vn

Submitted 02 July 2026; Revised 10 October 2026; Accepted 20 October 2026; Available online 26 October 2026.

Copyright © 2026 The Authors.

Abstract: This study presents a three dimensional Computational Fluid Dynamics (CFD) investigation of smoke propagation and thermal behavior in an underground parking garage equipped with a jet fan based mechanical ventilation system during a single representative multi vehicle fire event, in which three motorcycles ignite in sequence in a car park shared by cars and motorcycles. The simulation is implemented using the open source platform OpenFOAM, utilizing the *buoyantPimpleFoam* solver to incorporate buoyancy driven flow, heat transfer, and passive scalar smoke transport. The geometric configuration is based on a full scale underground parking facility, including structural elements, ventilation ducts, jet fans, and defined volumetric heat release zones to represent the fire source. Turbulence effects are captured using the *k- ω* SST model to simulate complex flow separation and recirculation induced by jet fans. The simulation results provide detailed temporal and spatial distributions of velocity, temperature, and smoke concentration. The temperature above the fire reaches a peak of 155.1 °C, whereas at the breathing height of 1.7 m the 60 °C limit is exceeded only in the immediate vicinity of the burning motorcycles. The floor area covered by dense smoke (normalised concentration $\phi > 0.1$) peaks at 1.51% at $t = 150$ s and falls to 0.01% at $t = 270$ s, while the smoke concentration at the entrance remains below 0.4% of the source value. Quantitative analysis demonstrates that the ventilation system enhances smoke transport toward exhaust ducts, mitigates overall smoke accumulation within the parking space, and maintains the selected thermal and smoke criteria for the duration of the considered fire event. The proposed workflow provides a reproducible framework for evaluating jet fan configurations and smoke control strategies, focusing on ventilation performance and smoke transport rather than detailed combustion modeling.

Keywords: CFD; Jet fan; OpenFOAM; Smoke control; Underground parking.

1. INTRODUCTION

Rapid urbanisation worldwide has intensified pressure on available land in densely populated metropolitan areas. As surface level space becomes increasingly constrained, underground construction has emerged as a strategic response across many countries, encompassing car parks, transit stations, utility corridors, and commercial facilities. Underground car parks, in particular, represent one of the most common applications of subsurface space in modern urban developments. The global transition towards electric vehicles, motivated by climate change mitigation and air quality improvement goals, has introduced a new category of fire risk in enclosed parking facilities. Lithium ion battery thermal runaway events can produce prolonged and intense heat release, large volumes of toxic smoke, and combustion by products that are difficult to suppress with conventional fire fighting approaches [1, 2]. Recent full-scale fire tests and performance-based safety analyses of electric and alternative fuel vehicles in underground car parks have confirmed these hazards [3–5]. Simultaneously, residual CO, CO₂, and NO_x emissions from internal combustion engine vehicles continue to degrade air quality during normal operation [6]. The confined geometry and limited natural ventilation of underground structures amplify these hazards, making fire safety and air quality management critical considerations from the earliest stages of design.

Mechanical ventilation systems particularly impulse ventilation systems (IVS) employing jet fans have become the preferred solution for smoke and pollutant control in underground car parks worldwide. By generating a directed airflow without the need for extensive ductwork, jet fan systems offer flexibility in layout and can be adapted to a wide range of structural configurations [7, 8]. However, smoke transport in underground spaces is a complex three dimensional phenomenon governed by the interaction of forced convection, thermal buoyancy, turbulent mixing, and geometric constraints [9, 10]. Accurately predicting system performance under fire conditions therefore requires tools capable of resolving these coupled effects. Computational Fluid Dynamics (CFD) simulation has established itself as the primary method for this purpose, enabling detailed analysis of

Table 1. Geometric and design specifications of the underground car park.

Parameter	Specification / Description
Maximum length	120 m
Maximum width	70 m
Height	5 m
Clear height	4.2 m
Car park floor area	731.48 m ²
Volume	3827.18 m ³
Entrance door	8 m (width) × 4 m (height)
Columns	7 units of 1×3 m (horizontal spacing: 19 m, longitudinal spacing: 8 m) 4 units of 2×1 m (horizontal spacing: 10 m, longitudinal spacing: 10 m)
Vehicles	3 car models, 6 motorcycle models (parked, non-burning obstacles)
Vent ducts	4 units
Jet fans	5 units (1.2 m diameter, 2.5 m length): 4 fans placed transversely in the main bay (fan1) 1 fan placed longitudinally near the entrance bay (fan2)

velocity, temperature, and smoke concentration fields without costly full scale physical tests; it has also been used to analyse the smoke temperature and the distribution of smoke gas components in buildings protected by mechanical smoke control systems such as staircase pressurisation [11].

CFD has been widely applied to the analysis of ventilation and smoke control in underground car parks. Lu et al. [12] pioneered the use of numerical simulation to compare impulse ventilation systems with traditional ducted ventilation, establishing that jet fan configuration and exhaust flow rate are primary determinants of smoke control performance. Deckers et al. [13, 14] further advanced this field by conducting extensive CFD simulations and full scale experiments; their parametric analysis quantified the effects of heat release rates and building geometry on smoke control effectiveness. Further studies have expanded the scope of CFD based investigation into specific operational scenarios. Nugroho et al. [15] utilized CFD modeling to map velocity fields and smoke distribution, specifically assessing how varying ventilation arrangements affect smoke control in underground garages. Similarly, Çakır and Ün [16] applied ANSYS Fluent to simulate an impulse ventilation system in a large underground car park, providing a detailed comparative analysis of different jet fan array layouts on smoke clearance times. More recent CFD studies have assessed the ventilation of car parks under fire conditions [17], induced ventilation systems in underground garages [18], the effect of the fan orientation on the smoke exhaust efficiency [19], and the data-driven optimisation of fan placement using CFD-generated data [20]; a recent review summarised the state of the art and the open challenges, in particular model validation and the representation of the fire scenario [21]. Beyond these studies, the works in [22, 23] have provided fundamental insights into how longitudinal ventilation velocities and structural elements affect the fire induced flow temperature and smoke control. Most existing studies rely on commercial software or FDS. While well validated, their high licensing costs and closed source nature present barriers for researchers seeking to develop customised modelling approaches [24]. The open source CFD platform OpenFOAM has attracted interest as a flexible alternative, and it has recently been applied to smoke dispersion in enclosed compartments [25] and to pool fires in large under-ventilated environments [26]. It is supported by efficient RANS based turbulence closures like $k-\omega$ SST which provide reliable flow profiles at lower computational costs than LES [27]. Concurrently, ensuring clear evacuation paths by keeping the conditions within the limits that people can withstand [9, 28] remains the absolute metric for verifying system efficacy.

The studies above almost exclusively consider car fires in car parks designed for cars. In Vietnam and other South-East Asian countries, however, motorcycles are the most common private vehicle, and the basements of commercial and residential buildings are usually shared by cars and many motorcycles, so a fire involving a few motorcycles is a realistic design scenario. This kind of case has not received much attention so far. The contribution of the present work is therefore a CFD assessment of a jet fan ventilation system for a fire of three motorcycles that ignite one after another in a car park shared by cars and motorcycles, using an open source and reproducible OpenFOAM workflow in which the jet fans follow a fan characteristic curve and their operating points are reported, together with a quantitative, time-resolved evaluation of the thermal and smoke conditions at the breathing height.

In this paper, we present a CFD simulation of smoke propagation, temperature distribution, and smoke control effectiveness of a jet fan ventilation system in an underground car park during a fire event, implemented entirely within OpenFOAM. We establish a complete simulation workflow covering geometry modelling, mesh generation, boundary condition specification,

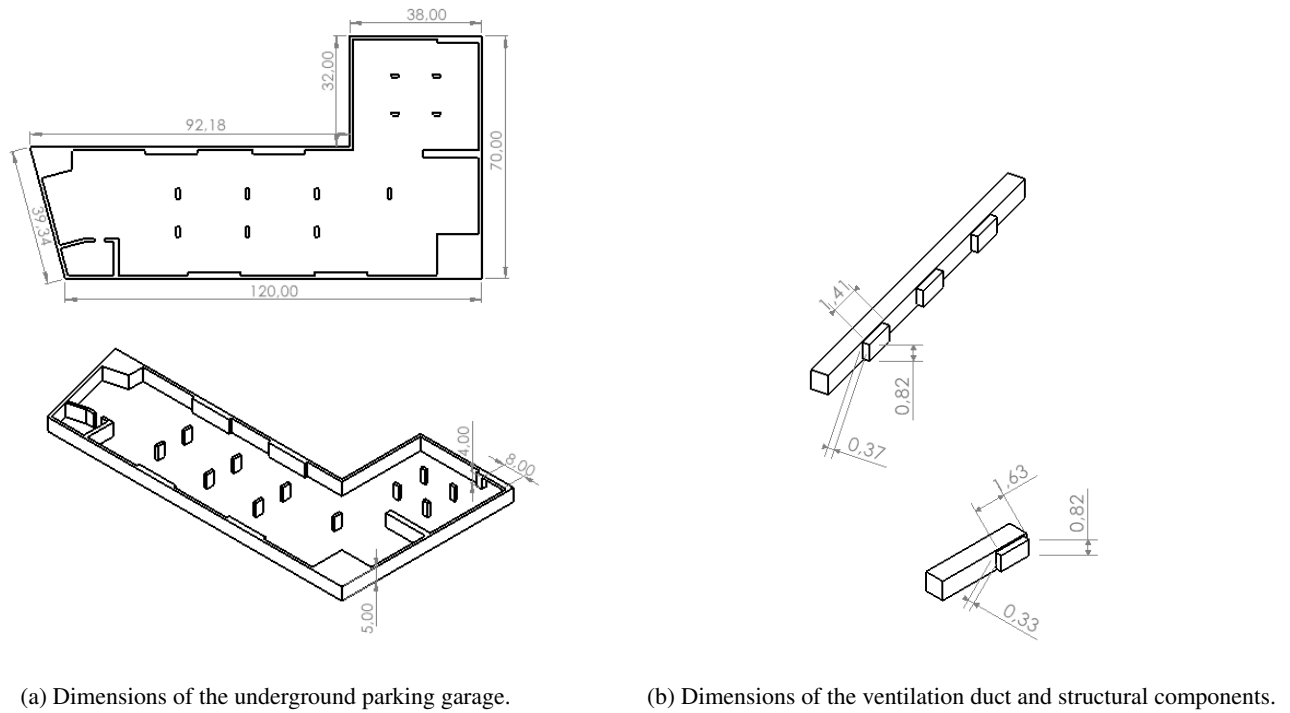


Figure 1. Geometric design specifications of the car park: (a) overall garage dimensions and (b) ventilation duct details.

turbulence and heat transfer modelling, and fire source representation, through to post processing of smoke and temperature fields. The study aims to demonstrate the suitability of OpenFOAM as a cost effective and flexible tool for the design and performance evaluation of smoke control ventilation systems in underground car park facilities.

2. GEOMETRY AND BOUNDARY CONDITIONS

The geometric model is inspired by a representative underground parking layout from a commercial building project. However, the computational domain has been simplified to focus on key ventilation and smoke propagation characteristics. The detailed geometric and design specifications of the underground car park are summarized in Table 1. The overall layout and main dimensions of the parking garage are presented in Figure 1a, while specific details regarding the ventilation duct system and structural components are provided in Figure 1b. Furthermore, the three dimensional computational representation of the parking environment, including the configuration of columns and ventilation equipment, is shown in Figure 2.

Table 2. Ventilation system details for the fire simulation.

Parameter	Simulation value
Air supply inlet (inlet)	$\mathbf{U} = (0, 0, 2) \text{ ms}^{-1}$
Air supply temperature	26.85°C
Exhaust opening (vent_duct_exhaust)	$\dot{m} = 15 \text{ kg s}^{-1}$
Active jet fans	fan1, fan2
Air supply mode	Mechanical
Air extraction mode	Mechanical

2.1 Mesh Generation

The domain is discretised using OpenFOAM's finite volume method, starting with a background hexahedral mesh (blockMesh: $182 \times 7 \times 98$ cells). Feature edges are extracted via surfaceFeatureExtract to preserve geometric accuracy, and the final mesh (Figure 3) is generated using snappyHexMesh. Refinement levels of 2 and 3 are applied to structural surfaces and jet fan regions, respectively, to accurately resolve high-velocity gradients. Jet fan faces are configured as internal patches for fan model boundary conditions. To maintain computational efficiency, boundary layer meshing is omitted, resulting in a total of 5.93×10^5 cells (Figure 3). Mesh quality—verified via checkMesh against non-orthogonality, skewness, and aspect ratio criteria—confirms suitability for resolving the intended thermal and smoke transport fields.

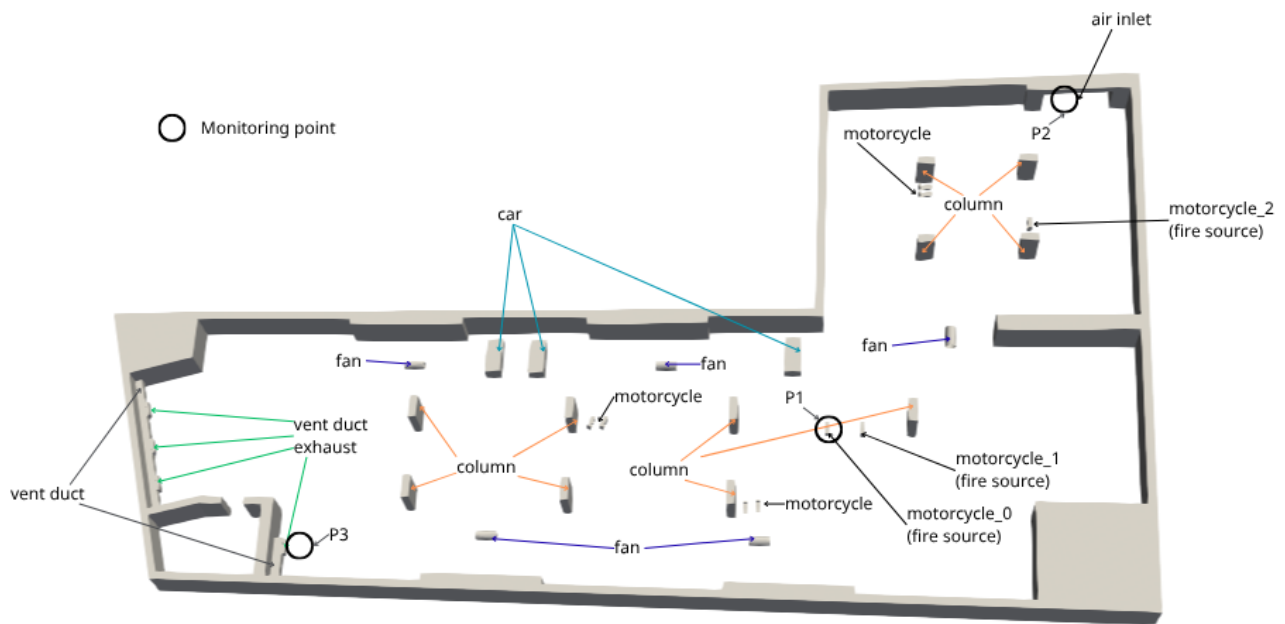


Figure 2. Three dimensional model of the underground car park.

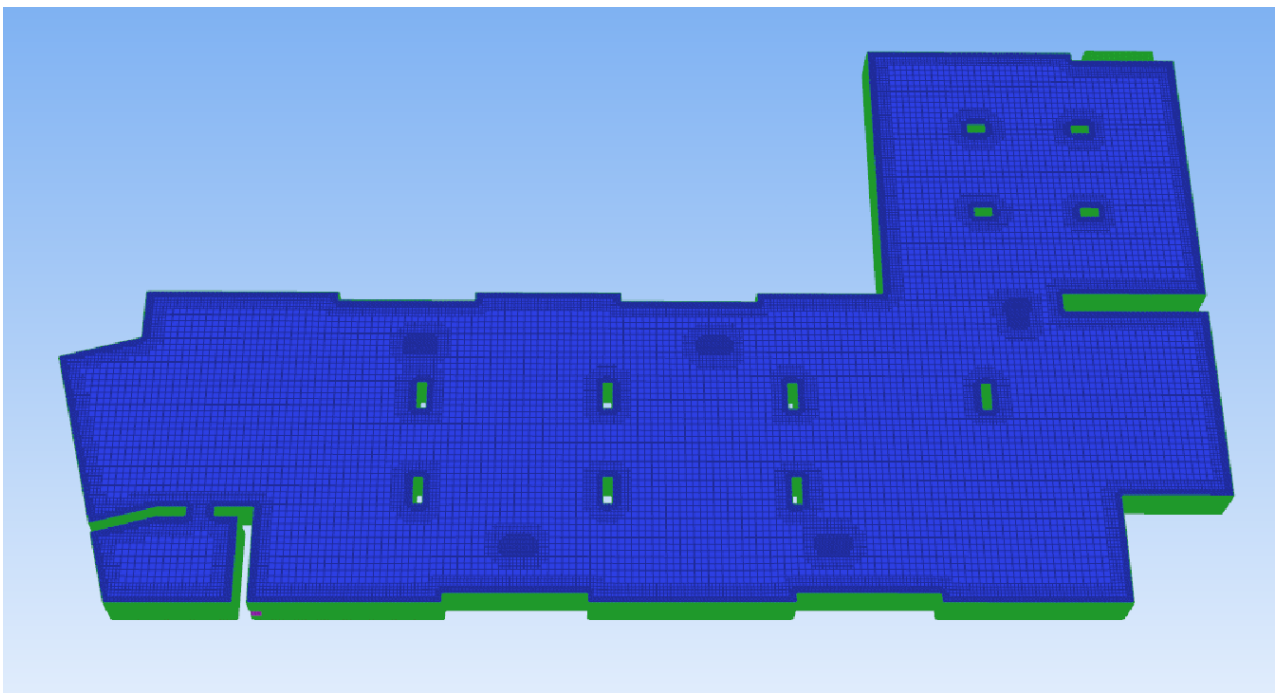


Figure 3. Three dimensional computational mesh of the underground car park (593,050 cells).

2.2 Boundary Conditions

Fresh air is supplied at 2 m/s and 26.85 °C through the entrance, with parameters detailed in Table 2. Smoke extraction is modelled at the far-end ducts using a fixed mass flow rate of 15 kg/s, utilizing `flowRateOutletVelocity` and `inletOutlet` boundary conditions to manage potential reverse flow. As detailed in Table 2, jet fans are modelled via cyclic patches and characteristic pressure-jump curves; the fan characteristic and the operating points obtained in the simulation are given in Table 3. Stationary surfaces employ no-slip conditions and standard wall functions for turbulence. Fire sources (`Motorcycle_0`, `Motorcycle_1`, `Motorcycle_2`): three volumetric cell zones in which the temperature is fixed at 276.85 °C and the smoke concentration is fixed at $\phi = 1$ during their activation periods (Table 4), using `fvOptions` constraints. Their function is to act as heat and smoke sources: the hot gas generates the buoyant plume and the smoke is then transported by the flow field. This simplified heat source approach ensures numerical stability for ventilation assessment without requiring explicit combustion modelling. Initial domain conditions, including ambient temperature and pressure, are summarized in Table 4.

Table 3. Jet fan parameters used in the simulation and their operating points.

Type	Quantity	Operating parameters
Axial jet fan (cyclic patch)	5 units in 2 cyclic zones (fan1: 4 units, fan2: 1 unit)	Pressure jump Δp defined by a characteristic curve (jumpTable) as a function of the mean face-normal velocity U_n , decreasing linearly from $\Delta p_{\max} = 350$ Pa at $U_n = 0$.
Operating point, fan1	4 units	$\Delta p = 350$ Pa; $Q \approx 80\text{--}83$ m ³ /s in total (≈ 20 m ³ /s per fan); $U_n \approx 18.5$ m/s
Operating point, fan2	1 unit	$\Delta p \approx 338$ Pa; $Q \approx 18.1\text{--}18.8$ m ³ /s; $U_n \approx 16.1$ m/s

Table 4. Fire simulation input parameters.

Parameter	Value
Domain dimensions	131.5 × 70.35 × 5.05 m
Ambient temperature	26.85 °C
Fire sources	3 burning motorcycles (Motorcycle_0, Motorcycle_1, Motorcycle_2)
Source zone volume	≈ 1.12 m ³
Source temperature	276.85 °C
Heat source activation, Motorcycle_0 & _1	0 – 150 s
Heat source activation, Motorcycle_2	30 – 210 s
Smoke source ($\phi = 1$), Motorcycle_0 & _1	0 – 190 s
Smoke source ($\phi = 1$), Motorcycle_2	60 – 270 s
Smoke diffusion	1.0×10^{-5} m ² s ⁻¹
Solver	buoyantPimpleFoam
Turbulence model	$k\text{-}\omega$ SST
Mesh cells	593,050
Duration	270 s

The fire growth profile, presented in Figure 4, illustrates the dynamic heat release rates (HRR) imposed on the simulation. The synchronized activation of the motorcycles generates a staggered peak in thermal loading, which tests the responsiveness of the ventilation system. As shown in Figure 4, the HRR peaks align with the established ignition sequence, providing a robust boundary condition for evaluating buoyancy induced flow. The three cars and the six other motorcycles are not burning and act only as obstacles to the flow; the scenario is a single representative design case rather than a reconstruction of a particular fire event.

3. COMPUTATIONAL FLUID DYNAMICS

The Computational Fluid Dynamics (CFD) methodology is predicated on discretising the flow domain into a finite number of small control volumes (mesh cells). Within each of these cells, the governing conservation equations for mass, momentum, and energy—collectively constituting the Navier-Stokes system—are discretised and solved through an iterative numerical procedure until a converged solution is achieved. This analytical framework facilitates the high-fidelity prediction of critical environmental parameters within underground parking facilities, including temperature fields, velocity distributions, and temporal smoke concentration gradients, all without the necessity of expensive full-scale experimental validation during the design phase. By analysing the simulation results, it is possible to rigorously assess the effectiveness of the ventilation strategy, the performance of the smoke extraction system, and the influence of structural obstructions on airflow guidance and smoke containment. In the context of an underground car park fire simulation, the CFD model integrates the fundamental physical phenomena governing the fire event, including heat release, smoke transport and diffusion, and the influence of mechanical ventilation—comprising fresh air intake, extraction systems, and jet fans. The simulation outputs provide comprehensive insight into smoke propagation pathways and thermal stratification at discrete time intervals throughout the fire development, which are vital for judging whether occupants can evacuate safely. The simulation workflow is structured into the following systematic stages:

1. Geometric Modelling: Construction of the parking facility's geometry and definition of the computational domain boundaries.
2. Model Export: Preparing the geometric data for mesh generation.
3. Mesh Generation: Discretisation of the domain into a high-quality computational mesh capable of resolving localized flow gradients.
4. Specification of Conditions: Defining the physical models, boundary conditions, and initial states for all structural surfaces,

air supply inlets, extraction ducts, jet fans, and fire source zones.

5. Numerical Discretisation and Solution: Implementing the discretised governing equations and performing the time-accurate numerical integration of the flow field.
6. Convergence and Reliability Analysis: Monitoring the stability of the numerical solution and ensuring the validity of the simulation results.
7. Post-processing and Visualisation: Analysing the data to extract quantitative results and visualise the flow phenomena.

3.1 Governing Equations

The CFD analysis employs the Finite Volume Method (FVM) applied to a three-dimensional computational grid. The governing partial differential equations are transformed into their integral form over each control volume and solved using a solver suitable for compressible, buoyancy-driven flows involving heat transfer, which characterizes the conditions within a parking facility during a fire event. The specific physical parameters and underlying modelling assumptions adopted for this simulation are summarized in Table 5. To represent the smoke concentration, a normalized passive scalar ϕ (dimensionless; $\phi = 0$ for clean air and $\phi = 1$ for undiluted smoke at the source) is used, governed by an advection-diffusion equation with a specified diffusion coefficient D , thus quantifying the relative concentration of the smoke contaminant rather than explicitly resolving species mass fractions.

Table 5. Assumptions and physical parameters used in the simulation.

No.	Parameter / Assumption	Value
1	Gas model	Ideal gas
2	Specific heat capacity C_p	1010 J/(kg · K)
3	Dynamic viscosity μ	1.82×10^{-5} Pa · s
4	Prandtl number Pr	0.713
5	Molar mass of air M	29 g/mol
6	Gravitational acceleration $\mathbf{g}$	(0, −9.81, 0) m/s ²
7	Thermal radiation	Not considered
8	Motorcycle fire source	$T = 276.85$ °C at zones Motorcycle_0, Motorcycle_1, Motorcycle_2
9	Smoke model	Passive scalar smoke
10	Smoke diffusion coefficient D	1.0×10^{-5} m ² /s

The CFD simulation is governed by the conservation laws for compressible flow with heat transfer and buoyancy, implemented via the *buoyantPimpleFoam* solver. The mass conservation (continuity) equation is defined as:

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \mathbf{U}) = 0 \quad (1)$$

where ρ is the gas density, $\mathbf{U}$ is the velocity vector, and t is time. The momentum conservation is described by the Navier Stokes equation:

$$\frac{\partial (\rho \mathbf{U})}{\partial t} + \nabla \cdot (\rho \mathbf{U} \mathbf{U}) = -\nabla p + \nabla \cdot \boldsymbol{\tau} + \rho \mathbf{g} \quad (2)$$

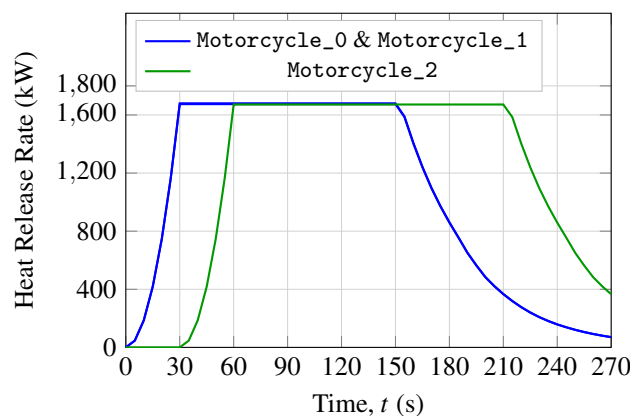


Figure 4. Imposed fire source evolution profile used in the simulation, showing the dynamic peaks of the fire sources; the growth phase follows the t -squared fire growth model [9].

where the effective stress tensor is given by $\boldsymbol{\tau} = (\mu + \mu_t)[\nabla \mathbf{U} + (\nabla \mathbf{U})^T - \frac{2}{3}(\nabla \cdot \mathbf{U})\mathbf{I}]$. In this solver, pressure is expressed as $p_{rgh} = p - \rho \mathbf{g} \cdot \mathbf{h}$ to enhance numerical stability for buoyancy driven flows, and the gas state is governed by the ideal gas law $p = \rho RT$, with $R = R_u/M$. Energy transport is computed using the conservation equation for sensible enthalpy $h = C_p T$:

$$\frac{\partial(\rho h)}{\partial t} + \nabla \cdot (\rho \mathbf{U} h) = \nabla \cdot (\alpha_{eff} \nabla h) + \frac{Dp}{Dt} \quad (3)$$

where $\alpha_{eff} = (\mu + \mu_t)/Pr_{eff}$ is the effective thermal diffusivity. Finally, smoke concentration ϕ is modelled as a passive scalar through the convection diffusion equation:

$$\frac{\partial \phi}{\partial t} + \nabla \cdot (\mathbf{U} \phi) = \nabla \cdot (D \nabla \phi) + S_\phi \quad (4)$$

where $D = 1.0 \times 10^{-5} \text{ m}^2/\text{s}$ and S_ϕ represents the smoke generation source term within the fire zones.

3.2 Turbulence Model

Due to the strongly turbulent nature of airflow in the underground car park during a fire characterised by flow separation, vortical structures around the jet fans, and large temperature gradients the Reynolds Averaged Navier Stokes (RANS) approach is employed to capture the influence of turbulence on the velocity, temperature, and smoke propagation fields at reasonable computational cost.

In this study, the Shear Stress Transport k - ω SST model (kOmegaSST) is selected. This model combines the advantages of the k - ω model in the near wall region and the k - ϵ model in the free stream region away from walls, using blending functions F_1 and F_2 to transition between them. The k - ω SST model has been widely applied in simulations of tunnel and car park ventilation and jet fan systems. The turbulence effects are modeled using the Shear Stress Transport k - ω SST model, which captures flow separation and recirculation through the transport equations for turbulent kinetic energy k and specific dissipation rate ω :

$$\frac{\partial(\rho k)}{\partial t} + \nabla \cdot (\rho \mathbf{U} k) = P_k - \beta^* \rho k \omega + \nabla \cdot [(\mu + \sigma_k \mu_t) \nabla k] \quad (5)$$

$$\frac{\partial(\rho \omega)}{\partial t} + \nabla \cdot (\rho \mathbf{U} \omega) = \frac{\gamma}{\mu_t} P_k - \beta \rho \omega^2 + \nabla \cdot [(\mu + \sigma_\omega \mu_t) \nabla \omega] + 2(1 - F_1) \frac{\rho \sigma_\omega^2}{\omega} \nabla k \cdot \nabla \omega \quad (6)$$

The turbulent viscosity is calculated as $\mu_t = \rho \frac{a_1 k}{\max(a_1 \omega, S F_2)}$, where $a_1 = 0.31$, S is the velocity strain rate magnitude, and F_2 is the SST limiter function, while the production term is bounded by $P_k = \min(\mu_t S^2, 10 \beta^* \rho k \omega)$. For boundary conditions, standard wall functions `kqRWallFunction` and `omegaWallFunction` are applied at solid surfaces. At the air supply inlet and exhaust openings, turbulence intensity $I = 5\%$ and mixing length $L = 0.001 \text{ m}$ are utilized to specify k and ω values, with domain wide initial conditions of $k = 0.01 \text{ m}^2/\text{s}^2$ and $\omega = 1 \text{ s}^{-1}$. This approach ensures reliable prediction of flow reattachment and pressure gradients in enclosed spaces with lower computational cost compared to LES or DNS.

3.3 Numerical Settings

The governing equations are discretised using the Finite Volume Method. The numerical settings for the simulation are summarised in Table 6. The pressure velocity coupling is handled using the PIMPLE algorithm, which provides robust convergence for transient, buoyancy driven flows.

The numerical simulations were executed on a high performance workstation equipped with an Intel Core i9-14900K processor and 32 GB of RAM. To accelerate the computational process, the computational domain was decomposed, and the simulation was executed in parallel using 16 processor cores via OpenFOAM's Message Passing Interface (MPI) implementation. For the present grid comprising approximately 593,050 cells and a physical simulation duration of 270 s, the total computational wall clock time required to achieve the converged solution was approximately 33 hours.

Table 6. Numerical discretisation schemes and solver settings.

Category	Scheme / Solver
Time integration	Euler (first order implicit)
Gradient terms	Gauss linear
Divergence terms	Gauss upwind (default), Gauss linear for stress
Laplacian terms	Gauss linear corrected
Pressure solver (p_{rgh})	GAMG with GaussSeidel smoother
Momentum (U) & Energy (h)	PBiCGStab with DILU preconditioning

The PIMPLE algorithm employs an outer corrector loop to maintain stability for highly transient fire simulations. For the pressure equation (p_{rgh}), a Geometric Algebraic Multi Grid (GAMG) solver is employed, while the momentum (U), energy (h), and turbulence transport equations (k , ω) are solved using the Preconditioned Bi Conjugate Gradient Stabilised (PBiCGStab)

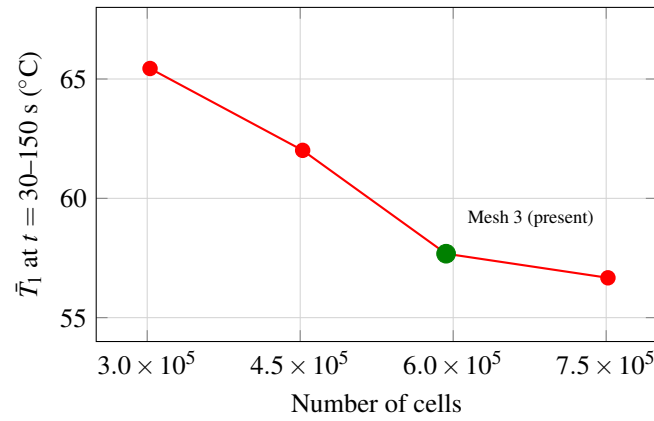


Figure 5. Grid independence test: temperature at Point 1 averaged over $t = 30-150$ s for the four meshes.

solver. Convergence is monitored through residual controls for all fields ($p_{rgh}, U, h, k, \omega, \phi$), set to a tolerance of 10^{-4} with a relative tolerance of 0.0, ensuring high fidelity results throughout the simulation period. The pressure velocity coupling is governed by the PIMPLE loop with one outer corrector and one non orthogonal corrector to handle grid skewness from snappyHexMesh.

3.4 Grid Independence Test

A grid independence test was carried out to check how much the results depend on the mesh size. Four meshes were generated with the same meshing procedure, with 302 776, 452 409, 593 050 and 751 734 cells for Meshes 1 to 4, respectively. Mesh 3 is the mesh used for all the results presented in this paper. All four cases were run with identical boundary conditions, physical models, numerical settings and simulated time. The quantity compared is the temperature at Point 1 above the fire (Section 4.1). Because the instantaneous temperature fluctuates in time, the value used is the time average over $t = 30-150$ s, when all three motorcycles are burning, $\bar{T}_1$.

Figure 5 shows $\bar{T}_1$ for the four meshes: 65.4, 62.0, 57.7 and 56.7 °C for Meshes 1 to 4, respectively. The value decreases as the mesh is refined, and the change becomes smaller: from Mesh 3 to Mesh 4 it is only 1.0 °C. Taking the temperature rise above the ambient temperature (26.85 °C), Mesh 3 differs from the finest Mesh 4 by 3.4%, whereas Meshes 2 and 1 differ by 18% and 29%. Mesh 3 (593 050 cells) was therefore kept as a compromise between accuracy and computational cost.

3.5 Model Validation

Full-scale experimental data for the underground car park considered in this study are very difficult to obtain. Therefore, the numerical model was validated against a comparable experiment of Ampofo and Karayiannis [29]. The experiment concerns turbulent natural convection of air in a square cavity 0.75 m high, 0.75 m wide and 1.5 m deep, with the vertical walls kept at 50 °C (hot wall) and 10 °C (cold wall). The validation case was solved using the same mathematical model as the car park simulation: the ideal-gas air properties listed in Table 5, the $k-\omega$ SST turbulence model with the same wall treatment, and the same discretisation schemes listed in Table 6. The smoke was computed with exactly the same smoke model as in the car park, i.e. a passive scalar ϕ with a diffusion coefficient $D = 1 \times 10^{-5}$ m²/s (Eq. (4)).

The experiment has no smoke measurements, so the smoke model was validated in an indirect way. Heat and a passive scalar follow the same type of transport equation. We therefore set $\phi = 1$ on the hot wall, $\phi = 0$ on the cold wall, and ϕ equal to the measured dimensionless temperature on the top and bottom walls. With these conditions, ϕ should match the dimensionless temperature $\theta = (T - T_c)/(T_h - T_c)$, so the computed ϕ can be compared directly with the measured θ . The computational domain is two-dimensional with 160×160 cells, refined towards the walls. The temperatures on the top and bottom walls, as well as all other conditions, follow the experiment [29].

Figure 6 compares the computed results with the measurements at mid-height of the cavity ($Y = y/L = 0.5$) near the hot wall, where $X = x/L$ is the distance from the hot wall divided by the cavity width $L = 0.75$ m, and y is the height above the bottom wall. Overall, the simulation follows the measured trends well. The vertical velocity v is divided by the buoyancy velocity $V_0 = \sqrt{g\beta H\Delta T}$, where $g = 9.81$ m/s² is the gravitational acceleration, β is the thermal expansion coefficient of air, $H = 0.75$ m is the cavity height and $\Delta T = T_h - T_c = 40$ °C is the wall temperature difference, giving $V_0 \approx 1$ m/s as in [29]. For the velocity (Figure 6a), the model reproduces the upward flow along the hot wall: the maximum velocity is at $X \approx 0.0064$ in the simulation and at $X \approx 0.0067$ in the experiment, while its value ($v/V_0 = 0.258$) is about 21% higher than the measured one (0.213). The simulation agrees well with the measurements very close to the wall ($X < 0.005$), but the velocity drops to zero faster further away ($0.02 < X < 0.05$). For the smoke (Figure 6b), the model reproduces well the fall of the concentration in the thin layer next to the wall ($X < 0.005$). Further away, the computed concentration falls faster than the measurements, with the largest difference of about 0.13 for $0.009 < X < 0.014$, where the computed curve has its minimum ($\phi \approx 0.44$ at $X \approx 0.014$); for $X > 0.03$, the computed value stays almost constant at $\phi \approx 0.47-0.48$, compared with measured values of $\theta \approx 0.50-0.54$. Overall, the present model reproduces the main features of both the velocity field and the smoke transport with acceptable

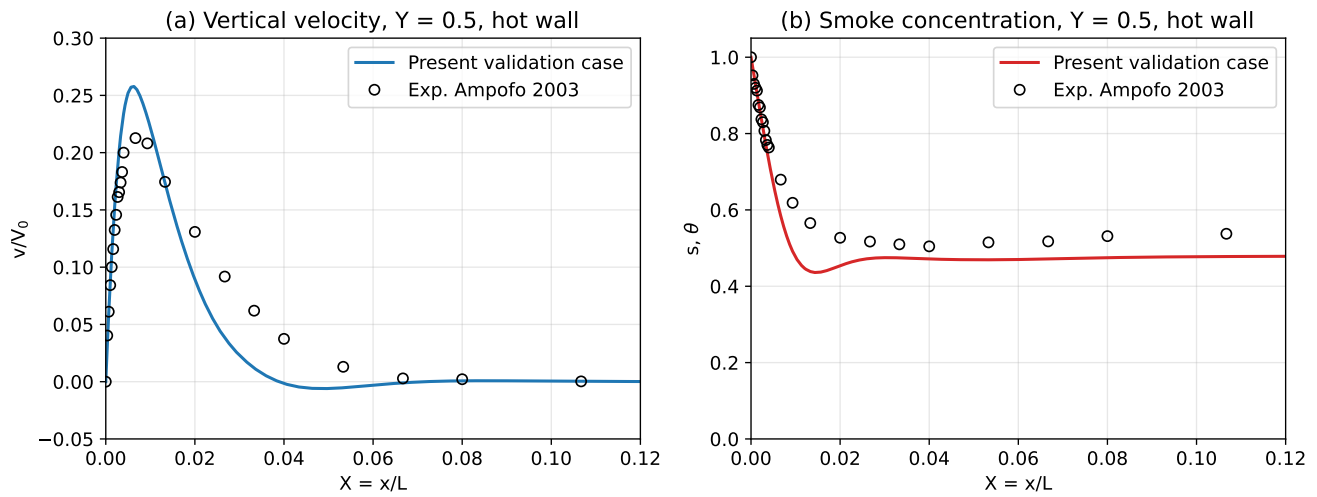


Figure 6. Comparison with the experiment of Ampofo and Karayiannis [29] at mid-height of the cavity ($Y = 0.5$) near the hot wall, as a function of the dimensionless distance from the hot wall $X = x/L$: (a) dimensionless vertical velocity v/V_0 , with the buoyancy velocity $V_0 = \sqrt{g\beta H\Delta T} \approx 1$ m/s; (b) smoke concentration ϕ of the present validation case compared with the measured dimensionless temperature θ .

accuracy, which confirms that it is suitable for the simulation of smoke and heat propagation in the underground car park.

4. RESULTS AND DISCUSSION

4.1 Localized Transient Thermal and Smoke Response

To quantitatively evaluate transient thermal and smoke propagation, virtual monitoring points (probes) were established using OpenFOAM's `postProcess` utility. These probes record data at three critical locations (Figure 2): Point 1 (Fire region) near the ceiling ($y = 4.8$ m, directly above `Motorcycle_0`) to capture peak thermal intensity; Point 2 (Air inlet) at the entrance ($y = 3.8$ m) to monitor ambient conditions and verify the suppression of thermal backlayering; and Point 3 (Exhaust vent) upstream of the duct ($y = 2.1$ m) to assess extraction efficiency.

Figure 7a illustrates the temperature evolution: Point 1 shows rapid thermal escalation to $\approx 155.1^\circ\text{C}$ at $t = 3$ s, followed by a sudden decrease to about 60°C at $t \approx 35$ s, after which it fluctuates between about 40 and 80°C until the heat sources of `Motorcycle_0` and `Motorcycle_1` are switched off at 150 s and then decreases to about 35°C . The initial peak and the sudden drop have the following explanation. The fire zones are switched on instantaneously at 276.85°C at $t = 0$, while the air in the car park is initially at rest. The hot gas therefore rises as an almost vertical plume and reaches Point 1, located directly above `Motorcycle_0`, within about 3 s. In the following seconds the ventilation flow develops: the air entering through the door and the jets of the fans create a cross-flow that tilts the plume and carries the hot gas towards the exhaust openings, and the plume entrains cooler ambient air. Point 1 is then no longer located in the core of the plume but at its edge, so that its temperature decreases although the fire source is still active. The smoke concentration at Point 1 (Figure 7b) shows the same behaviour, with a peak of $\phi \approx 0.53$ at $t = 3$ s followed by a decrease, which confirms that the drop is caused by the deflection and dilution of the plume and not by a change of the source. Point 2 remains near-ambient, confirming that the jet fan-induced airflow isolates the fresh air supply from heat dispersion. Point 3 shows a gradual increase, reflecting the progressive transport of thermally diluted gases toward the exhaust ducts.

Regarding the safety of the occupants, Figure 7b depicts the evolution of the normalized smoke scalar field ϕ (where $\phi = 0$ denotes clean air and $\phi = 1$ denotes the undiluted source): Point 1 records elevated concentrations near the fire sources, while Point 2 remains negligible throughout the event ($\phi < 0.0031$), validating that the longitudinal momentum induced by the jet fans effectively arrests upstream smoke propagation and secures the primary evacuation path. At the exhaust vent, Point 3 reaches a peak smoke scalar value of approximately 4% of the undiluted source concentration ($\phi \approx 0.04$), confirming that the extracted flow carries only heavily diluted smoke and demonstrating the overall extraction efficiency of the ventilation system.

4.2 Simulation Results Over Time

Table 7 presents the temporal variation of temperature and smoke propagation extent within the car park at different simulation time instants. The values are extracted directly from the temperature field T and the smoke scalar field `smoke` during the simulation using the `buoyantPimpleFoam` solver. The light and dense smoke area fractions are the percentages of the horizontal plane at the breathing height ($y = 1.7$ m) in which the normalised smoke concentration exceeds $\phi = 0.01$ and $\phi = 0.1$, respectively. Since ϕ is the smoke concentration relative to the undiluted smoke at the source, light smoke ($0.01 < \phi \leq 0.1$) corresponds to smoke diluted between 10 and 100 times, and dense smoke ($\phi > 0.1$) to smoke diluted less than 10 times. These two thresholds are relative indicators chosen in this study to distinguish strongly diluted smoke from concentrated smoke.

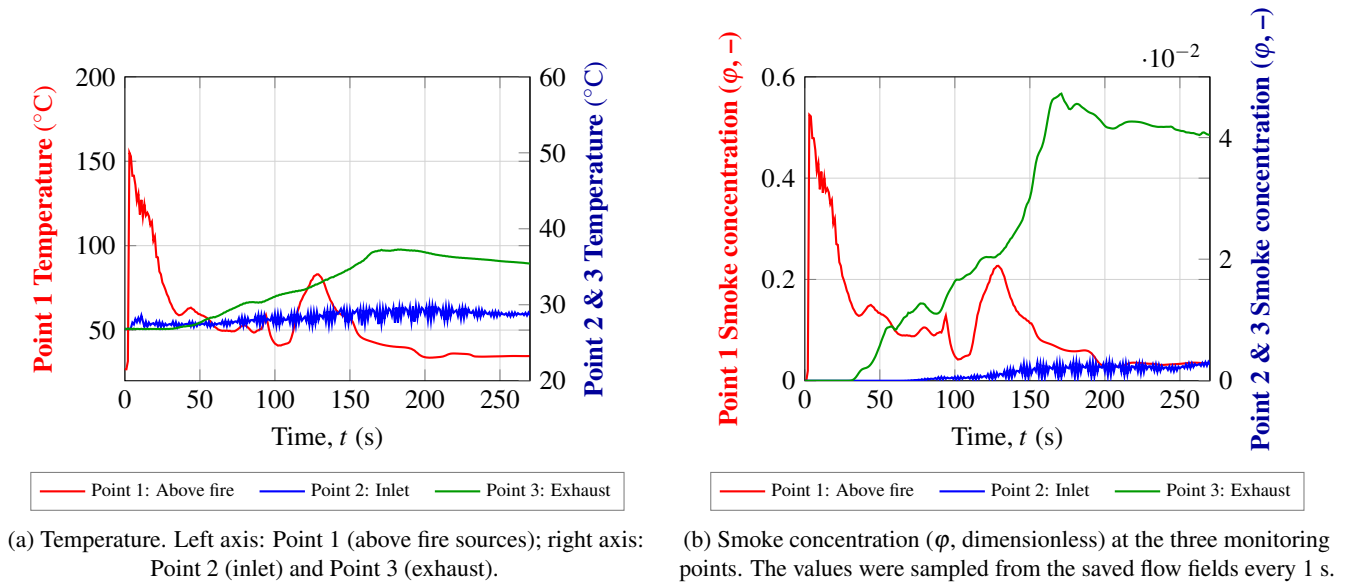


Figure 7. Transient profiles of (a) temperature and (b) smoke concentration at the three monitoring points.

Table 7. Temperature and smoke propagation results over time.

Time t (s)	Mean Temp. $\bar{T}$ ($^{\circ}\text{C}$)	Mean Smoke $\bar{\phi}$ (–)	Light smoke area fraction (%) ($\phi > 0.01$)	Dense smoke area fraction (%) ($\phi > 0.1$)
0	26.9	0.0000	0.00	0.00
30	27.6	0.0029	7.04	0.31
60	28.8	0.0073	18.80	0.63
90	30.7	0.0143	36.51	0.63
120	31.8	0.0188	50.93	0.71
150	33.0	0.0238	68.59	1.51
180	33.6	0.0281	83.83	1.10
210	33.8	0.0287	87.75	0.93
240	33.6	0.0278	90.09	0.03
270	33.5	0.0271	92.01	0.01

4.3 Analysis of Smoke Dynamics and System Efficacy

Figure 8 shows the transient smoke propagation at a breathing height of $y \approx 1.7$ m. At $t = 60$ s, smoke is localized near the burning motorcycles, keeping the remaining areas safe for evacuation. Through $t = 120$ s and $t = 180$ s, jet fans effectively direct the smoke plume toward the exhaust vents, ensuring safe air upstream. By $t = 210$ s and $t = 240$ s, while smoke accumulates in downstream corridors, the extraction system efficiently removes the contaminants. Finally, at $t = 270$ s, the ventilation strategy clears the residual dense smoke, demonstrating its effectiveness. These results highlight the dominant role of jet fan-induced momentum in controlling smoke movement. The forced airflow establishes a primary corridor that guides smoke toward exhaust ducts, preventing large-scale accumulation. Local flow disturbances near structural obstructions—such as columns and vehicles—create wake regions where smoke may trap, leading to localized concentration increases; this behavior is consistent with previous studies on confined space ventilation [9, 13]. During the early fire stage, strong smoke stratification occurs, with buoyant gases accumulating near the ceiling while lower regions stay safe. Table 8 summarizes the ventilation efficacy: the temperature at breathing height is maintained below the 60°C limit, except near the fire source at $t = 120$ s. Furthermore, while light smoke coverage reaches $\approx 92.01\%$ by the simulation's end, dense smoke is effectively mitigated, decreasing from $\approx 0.71\%$ at $t = 120$ s to $\approx 0.01\%$ at $t = 270$ s. Overall, the jet fan system maintains acceptable evacuation conditions, although its efficiency remains dependent on obstruction layout and fan positioning.

The criteria used in Table 8 and their basis are as follows. The temperature limit of 60°C at the breathing height is the limit for convective heat exposure used in performance-based assessments of car parks according to PD 7974-6 [3], and similar temperature limits are used in safety studies of underground transport facilities [28].

On this basis, the conditions can be considered safe for the occupants with respect to the evaluated criteria, for three reasons. First, at the breathing height the temperature exceeds 60°C only in the immediate vicinity of the burning motorcycles, i.e. in

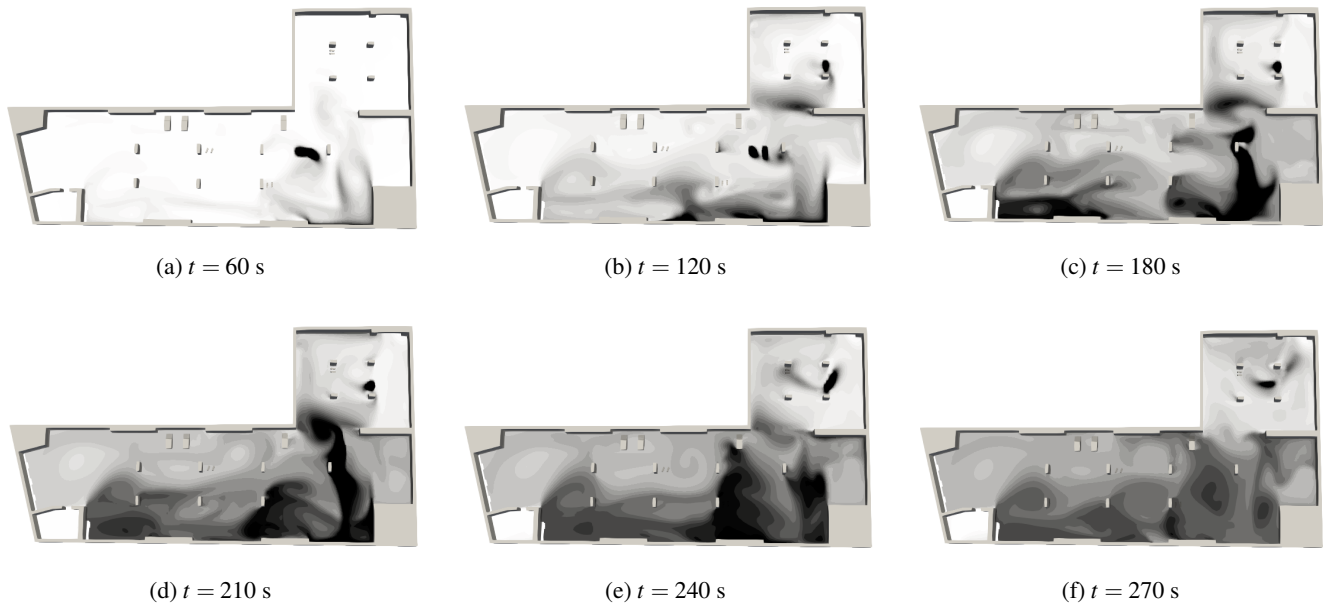


Figure 8. Smoke distribution (smoke) at height $y \approx 1.7$ m at different simulation time instants.

the area that the occupants leave first. Second, dense smoke ($\phi > 0.1$) never covers more than 1.51% of the floor area and is reduced to 0.01% at $t = 270$ s. Third, the main escape route through the entrance remains practically free of smoke and heat: at Point 2 the smoke concentration remains below 0.0031 (smoke diluted more than 300 times) and the temperature remains below 30 °C, in agreement with the requirement that ventilation keeps the escape routes upstream of the fire free of smoke [3, 8]. Light smoke ($\phi > 0.01$), however, covers 92% of the floor area at the end of the simulation, and toxic gases, visibility and radiant heat flux [30] were not evaluated. A complete assessment of how safe the light smoke region is would require a combustion model that provides the concentrations of CO, CO₂ and soot, which is planned for future work.

Note: $T_{\max} = 276.85^\circ\text{C}$ corresponds to the fire source zones. After $t > 210$ s, heat sources are deactivated, causing $T_{\max}$ to drop to $\approx 48^\circ\text{C}$.

Table 8. Assessment of the thermal and smoke conditions at the breathing height ($y \approx 1.7$ m) against the selected criteria.

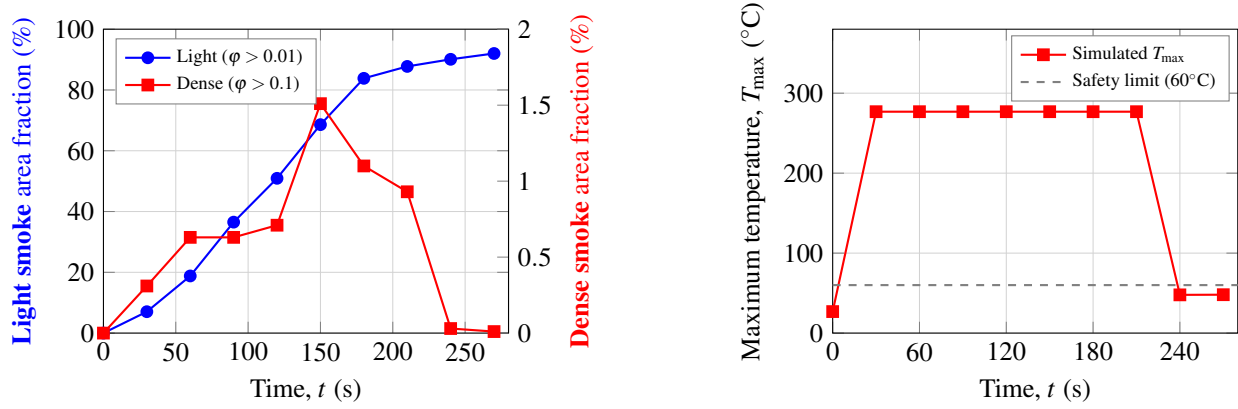
Criterion	Selected limit	$t = 120$ s	$t = 270$ s
Temperature at breathing height ($y \approx 1.7$ m)	$T < 60^\circ\text{C}$ [3]	Satisfied, except immediately adjacent to fire source	Satisfied
Dense smoke area	$\phi > 0.1$	$\approx 0.71\%$ of floor area	$\approx 0.01\%$ of floor area
Light smoke area	$\phi > 0.01$	$\approx 50.93\%$ of floor area	$\approx 92.01\%$ of floor area
Smoke at the entrance (Point 2)	–	$\phi < 0.001$	$\phi \approx 0.003$

The area based smoke concentration analysis in Figure 9a shows how safe the car park environment is. It is observed that while the "Light smoke" area increases progressively, reaching approximately 92% by $t = 270$ s, the "Dense smoke" area remains restricted to less than 2% throughout the entire event, as detailed in Figure 9a. This disparity confirms that the jet fan system effectively promotes dilution, preventing the formation of large scale hazardous zones.

The variation of maximum temperature within the domain, provided in Figure 9b, demonstrates the thermal regulation capability of the extraction system. The temperature exceeds the safety threshold of 60°C only in the immediate vicinity of the fire source zones, while the rest of the domain is maintained well within safe limits, as indicated by the trend in Figure 9b.

Figure 10 shows the temporal evolution of the mean smoke concentration ($\bar{\phi}$). Here, ϕ is a normalized passive scalar where 0 and 1 represent clean air and the undiluted source, respectively. The concentration rises steadily from $t = 0$ s to $t = 210$ s due to active smoke generation. Following the end of the smoke release from Motorcycle_0 and Motorcycle_1 at 190 s and the deactivation of the last heat source at 210 s, the concentration stabilizes and declines toward $t = 270$ s, although Motorcycle_2 still releases smoke, confirming that the mechanical ventilation system effectively extracts contaminants and prevents long-term accumulation.

Figure 11 shows the air velocity distribution at $t = 120$ s. The jet fan system creates a well-defined longitudinal flow corridor from the entrance to the exhaust ducts, with high-velocity zones (red and orange) confirming efficient ventilation. While structural elements like columns induce localized wake regions and minor flow separation, these disturbances remain contained



(a) Percentage of the car park area at breathing height ($y = 1.7$ m) covered by light and dense smoke. (b) Variation of maximum temperature ($T_{\max}$) inside the underground car park.

Figure 9. Smoke coverage and maximum temperature in the car park: (a) area fractions of light and dense smoke at breathing height and (b) maximum temperature $T_{\max}$.

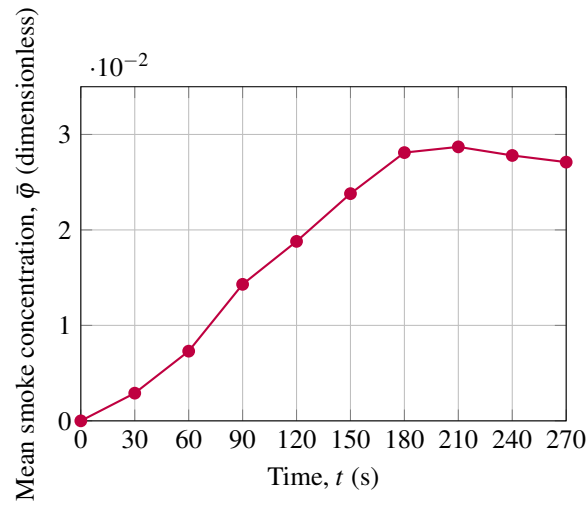


Figure 10. Temporal evolution of the mean smoke concentration ($\bar{\phi}$, dimensionless) throughout the underground car park.

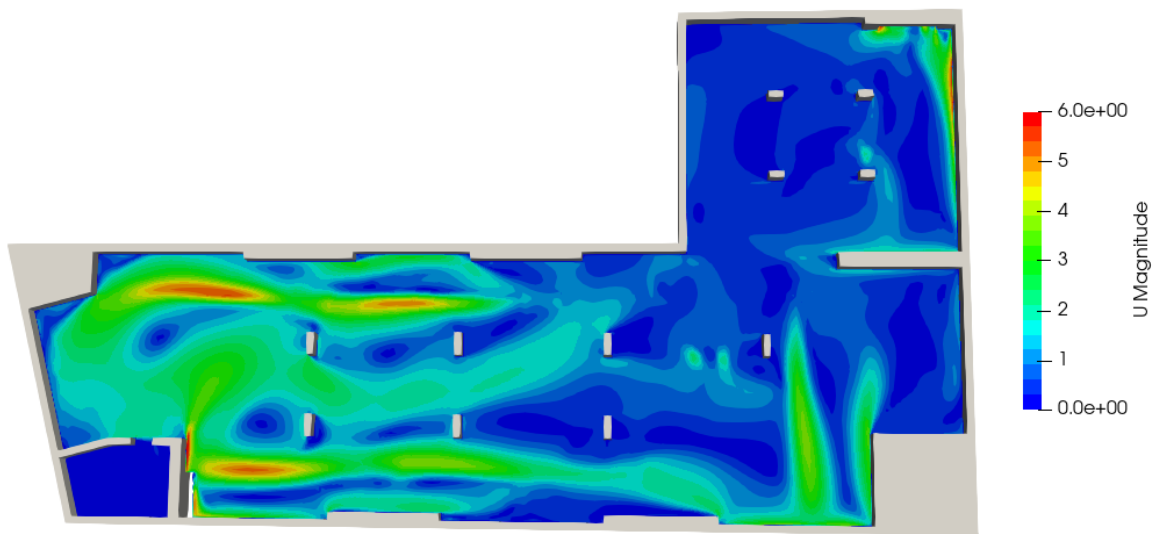


Figure 11. Air velocity contour at $t = 120$ s.

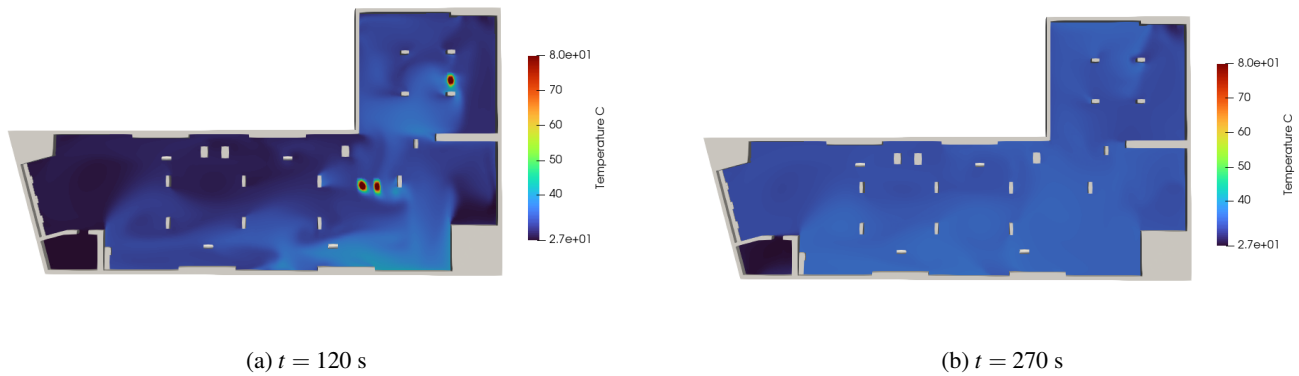


Figure 12. Temperature distribution at height $y = 1.7$ m.

and do not impede smoke clearance. This confirms the jet fan arrangement provides sufficient momentum to maintain consistent airflow throughout the facility.

The temperature distribution at a breathing height of $y \approx 1.7$ m is shown in Figure 12. At $t = 120$ s, localized high temperature regions are observed near the fire sources; however, the surrounding environment remains within safe thermal limits due to the convective cooling effect of the forced ventilation. By $t = 270$ s, following the cessation of the fire, the temperature across the car park returns to ambient levels. This thermal progression confirms that the ventilation system successfully prevents heat buildup, ensuring that temperature conditions at breathing height stay within safe limits throughout the fire event.

5. CONCLUSION

This study presented a comprehensive Computational Fluid Dynamics (CFD) investigation into smoke propagation and thermal distribution within a full-scale underground car park, evaluating the effectiveness of a jet fan-based impulse ventilation system during a multi-vehicle fire. The numerical framework was implemented within the open-source platform OpenFOAM, utilizing the *buoyantPimpleFoam* solver coupled with the $k-\omega$ SST turbulence model to accurately resolve the complex interactions between buoyancy-driven flows, heat transfer, and mechanical ventilation. The model was verified by a grid independence test, in which the temperature rise at Point 1 on the present mesh differed by 3.4% from that on the finest mesh, and validated against the natural convection experiment of Ampofo and Karayiannis, in which the location of the near-wall velocity maximum was reproduced within 5% and its magnitude within 21%. Simulation results provided a detailed spatio-temporal analysis of the fire event. Quantitative findings confirm that the jet fan configuration effectively establishes a primary flow corridor, successfully directing smoke toward designated exhaust ducts and preventing thermal backlayering at the air supply inlet. Monitoring critical parameters at the fire zone, air inlet, and exhaust vent demonstrated that the system maintains safe conditions for evacuation for most of the simulation. The temperature above the fire reached a peak of 155.1°C , while at the entrance the temperature stayed below 30°C and the smoke concentration below $\phi = 0.0031$. The jet fans operated at about $20\text{ m}^3/\text{s}$ per fan with a pressure jump of 338–350 Pa. Furthermore, the safety assessment at breathing height indicated that the ventilation strategy successfully confines hazardous conditions, despite minor recirculation zones near structural obstacles such as columns and parked vehicles. The 60°C limit was exceeded only next to the burning motorcycles, and the area covered by dense smoke ($\phi > 0.1$) peaked at 1.51% at $t = 150$ s and decreased to 0.01% at $t = 270$ s, although light smoke ($\phi > 0.01$) covered 92% of the floor area at the end of the simulation. The application of a volumetric heat source model proved to be a robust, efficient approach for ventilation-focused engineering assessments, achieving stable convergence without the computational overhead of complex combustion modelling. This study shows that the proposed OpenFOAM-based workflow is a reliable, cost-effective alternative to commercial CFD tools for performance-based fire safety design in underground parking structures. Future research will aim to enhance predictive accuracy by integrating more sophisticated combustion models and transient heat release profiles, which will also allow the evaluation of visibility and toxic gas concentrations. The present validation was performed against a laboratory-scale benchmark experiment; subsequent efforts should prioritize full-scale or reduced-scale car park fire experiments to provide further quantitative validation data. Furthermore, expanding the simulation scope to include diverse fire locations, varying ventilation intensities, and potential fan failure scenarios will contribute to a more holistic framework for optimizing fire safety in complex underground environments.

ACKNOWLEDGEMENT AND FUNDING

This research is funded by Thai Nguyen University, Vietnam, through the university-level science and technology project “Research on building a Digital Twin model based on numerical flow simulation software”.

DECLARATION OF CONFLICTING INTERESTS

The authors declare no potential conflicts of interest with respect to the research and publication of this article.

REFERENCES

- [1] P. Sun, R. Bisschop, H. Niu, and X. Huang, A review of battery fires in electric vehicles, *Fire Technology*, 56, 2020, 1361–1410.
- [2] F. Larsson, P. Andersson, P. Blomqvist, and B.-E. Mellander, Toxic fluoride gas emissions from lithium-ion battery fires, *Scientific Reports*, 7, 2017, 10018.
- [3] D. Brzezinska and P. Bryant, Performance-based analysis in evaluation of safety in car parks under electric vehicle fire conditions, *Energies*, 15, 2022, 649.
- [4] S. Kang, M. Kwon, J. Y. Choi, and S. Choi, Full-scale fire testing to assess the risk of battery electric vehicle fires in underground car parks, *Fire Technology*, 61, 2025, 4133–4163.
- [5] E. Leone and D. Papurello, Fire safety analysis of alternative vehicles in confined spaces: A study of underground parking facilities, *Fire*, 9, 2026, 20.
- [6] P. Sittisak, T. Charinpanitkul, and B. Chalermisinsuwan, Enhancement of carbon monoxide removal in an underground car park using ventilation system with single and twin jet fans, *Tunnelling and Underground Space Technology*, 97, 2020, 103226.
- [7] J. C. Viegas, The use of impulse ventilation for smoke control in underground car parks, *Tunnelling and Underground Space Technology*, 25, 2010, 42–53.
- [8] B. Merci and M. Shipp, Smoke and heat control for fires in large car parks: Lessons learnt from research?, *Fire Safety Journal*, 57, 2013, 3–10.
- [9] H. Ingason, Y. Z. Li, and A. Lönnemark, *Tunnel Fire Dynamics*, New York: Springer, 2015.
- [10] W. Wang, Z. Zhu, Z. Jiao, H. Mi, and Q. Wang, Characteristics of fire and smoke in the natural gas cabin of urban underground utility tunnels based on CFD simulations, *Tunnelling and Underground Space Technology*, 109, 2021, 103748.
- [11] M. A. Rusli, A. S. Ahmad Kamal, M. R. Tarmizi, M. F. K. Anwar Khan, H. Kamal, and M. A. Ismail, The smoke temperature analysis and smoke gas component distribution using the staircase pressurisation system, *Applications of Modelling and Simulation*, 9, 2025, 338–348.
- [12] S. Lu, Y. H. Wang, R. F. Zhang, and H. P. Zhang, Numerical study on impulse ventilation for smoke control in an underground car park, *Procedia Engineering*, 11, 2011, 369–378.
- [13] 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*, 57, 2013, 11–21.
- [14] X. Deckers, S. Haga, N. Tilley, and B. Merci, Smoke control in case of fire in a large car park: CFD simulations of full-scale configurations, *Fire Safety Journal*, 57, 2013, 22–34.
- [15] Y. S. Nugroho, B. Alianto, D. Nareshwara, and N. Astari, Modeling of smoke control in underground parking-garage fires, *International Journal of Technology*, 8, 2017, 1296–1305.
- [16] M. T. Çakır and Ç. Ün, CFD analysis of smoke and temperature control system of car park area with jet fans, *Journal of Engineering Research and Reports*, 13, 2020, 27–40.
- [17] R. Rahif and S. Attia, CFD assessment of car park ventilation system in case of fire event, *Applied Sciences*, 13, 2023, 10190.
- [18] Z. Wang, X. Zhou, X. Zhu, and J. Liu, Numerical simulation of the ventilation and fire conditions in an underground garage with an induced ventilation system, *Buildings*, 13, 2023, 2074.
- [19] T.-J. Lee, E.-Y. Oh, and H.-J. Song, Improved smoke exhaust efficiency through modification of ventilation fan orientation in underground parking lots, *Fire*, 8, 2025, 398.
- [20] M. Zandie, A. Nicsan, S. Sharifi, R. Marriot, M. Jaffar, K. Windows-Yule, and M. Jangi, Data-driven optimisation of the ventilation systems in underground car parks: a CFD-DNN study, *Journal of Building Engineering*, 98, 2024, 111181.
- [21] C. Stan, I. Năstase, F. Bode, and R. Calotă, Smoke and hot gas removal in underground parking through computational fluid dynamics: A state of the art and future challenges, *Fire*, 7, 2024, 375.
- [22] Y. Wu and M. Z. A. Bakar, Control of smoke flow in tunnel fires using longitudinal ventilation systems – a study of the critical velocity, *Fire Safety Journal*, 35, 2000, 363–390.
- [23] L. Li, S. Li, X. Wang, and H. Zhang, Fire-induced flow temperature along tunnels with longitudinal ventilation, *Tunnelling and Underground Space Technology*, 32, 2012, 44–51.
- [24] V. Novozhilov, Computational fluid dynamics modeling of compartment fires, *Progress in Energy and Combustion Science*, 27, 2001, 611–666.
- [25] Y. Zhao, H. Zhao, Z. Miao, D. Ai, and Q. Wang, A numerical study on the smoke dispersion and temperature distribution of a ship engine room fire based on OpenFOAM, *Sustainability*, 15, 2023, 15093.
- [26] M. Cavazzuti and P. Tartarini, Pool fires within a large under-ventilated environment: Experimental analysis and numerical simulation using OpenFOAM, *Fire Technology*, 60, 2024, 1891–1915.
- [27] F. R. Menter, Two-equation eddy-viscosity turbulence models for engineering applications, *AIAA Journal*, 32, 1994, 1598–1605.
- [28] C. H. Cheng, C. L. Chow, and W. K. Chow, A simulation study of tenability for passengers in a railway tunnel with arson fire, *Tunnelling and Underground Space Technology*, 108, 2021, 103679.
- [29] F. Ampofo and T. G. Karayiannis, Experimental benchmark data for turbulent natural convection in an air filled square

cavity, *International Journal of Heat and Mass Transfer*, 46, 2003, 3551–3572.

- [30] N. M. H. Nik Zulkifli, F. R. Munas, H. Abdul Hamid, M. K. Ishak, and M. A. Ismail, Performance of water cooling for radiation heat flux fuel storage tank, *ASEAN Engineering Journal*, 14, 2024, 23–28.