

Article

Not peer-reviewed version

CFD-Based Investigation of Smoke Evacuation Efficiency in a Hotel Fire Scenario: Antalya Case Study

[Ahmet Coşgun](#)^{*}, Ilkay Sayer, Ali Onur Gunduztepe, Ali Ufuk Dibic

Posted Date: 4 August 2025

doi: 10.20944/preprints202508.0236.v1

Keywords: fire installation; smoke evacuation; CFD analysis; NFPA standard



Preprints.org is a free multidisciplinary platform providing preprint service that is dedicated to making early versions of research outputs permanently available and citable. Preprints posted at Preprints.org appear in Web of Science, Crossref, Google Scholar, Scilit, Europe PMC.

Copyright: This open access article is published under a Creative Commons CC BY 4.0 license, which permit the free download, distribution, and reuse, provided that the author and preprint are cited in any reuse.

Disclaimer/Publisher's Note: The statements, opinions, and data contained in all publications are solely those of the individual author(s) and contributor(s) and not of MDPI and/or the editor(s). MDPI and/or the editor(s) disclaim responsibility for any injury to people or property resulting from any ideas, methods, instructions, or products referred to in the content.

Article

CFD-Based Investigation of Smoke Evacuation Efficiency in a Hotel Fire Scenario: Antalya Case Study

Ahmet Çoşgun ^{1,*}, İlkey Sayer ², Ali Onur Gunduztepe ² and Ali Ufuk Dibic ²

¹ Akdeniz University, Faculty of Engineering, Department of Mechanical Engineering, Antalya, Turkey

² Akdeniz University, Institute of Natural and Applied Science, Department of Mechanical Engineering, Antalya, Turkey

* Correspondence: acoskun@akdeniz.edu.tr

Abstract

Smoke generated during a fire consists of toxic gases and fine particles that pose significant risks to human health and safety. Effective smoke evacuation is therefore critical to ensure safe evacuation and minimize health hazards. In this study, a comprehensive engineering design and CFD-based performance evaluation of a smoke evacuation system was carried out for a five-star hotel conference hall in Antalya, Türkiye. The system was designed in compliance with NFPA 92 guidelines, including the calculation of air flow rates, duct sizing, and component selection. The physical configuration was modeled using AutoCAD, and detailed CFD simulations were performed using Simcenter STAR-CCM+ software. The 90-second fire scenario was simulated in 10-second intervals to capture the transient behavior of smoke dispersion and ventilation efficiency. The CFD model incorporated boundary conditions based on calculated fire heat release rates and ventilation velocities. Results show that the ventilation system effectively reduced smoke accumulation, maintained critical visibility levels at evacuation height, and controlled temperature rise. Although experimental validation through cold smoke tests could not be conducted, the simulation results align with international literature benchmarks. This study contributes to the development of high-fidelity, time-dependent CFD analysis methods for smoke management systems and provides practical insights for fire safety engineers and building designers.

Keywords: fire installation; smoke evacuation; CFD analysis; NFPA standard

1. Introduction

Fire safety continues to be one of the most critical design and operational concerns in complex buildings such as hotels, airports, shopping malls, and hospitals. These structures often host large numbers of occupants and contain multiple zones with diverse ventilation conditions, making emergency evacuation during a fire both a technical and human challenge. While fire suppression remains a vital component of fire protection strategies, modern fire engineering increasingly focuses on life safety through effective smoke control. Smoke evacuation systems are essential in reducing the risk of fatalities caused by inhalation of toxic gases—such as carbon monoxide (CO), hydrogen cyanide (HCN), and other harmful combustion products—while also maintaining visibility in escape routes and supporting the intervention of fire-fighting teams.

The presence of toxic smoke significantly impairs visibility and can lead to panic, delayed evacuation, and, ultimately, fatalities. Various studies have shown that most fire-related deaths are due not to burns but to smoke inhalation and the toxic effects of combustion byproducts [1–5]. Carbon monoxide and hydrogen cyanide, in particular, have been shown to cause irreversible damage to the nervous and cardiovascular systems even at low concentrations. Recent research has underscored the

importance of rapid mechanical smoke removal in reducing exposure duration, thereby improving the likelihood of survival [6–9].

Smoke control systems must be designed in accordance with recognized international standards and validated through experimental or computational methods. NFPA 92 provides detailed guidance for the design, calculation, and performance assessment of smoke management systems in various occupancy types [10]. Other regional standards, such as Türkiye’s National Fire Protection Guidelines, also establish technical thresholds for ventilation rates, toxic gas concentrations, and temperature gradients during fire events.

Computational Fluid Dynamics (CFD) simulations have become an indispensable tool in evaluating smoke propagation, system performance, and evacuation safety. Advanced CFD platforms such as STAR-CCM+ allow the modeling of complex, unsteady fire-induced flows using turbulence models (e.g., RNG k-ε, SST k-ω) and thermal buoyancy coupling, enabling time-resolved simulation of temperature, velocity, and gas concentration fields [11–18]. CFD not only supports system optimization but also serves as a digital testbed for performance validation, especially where physical testing is impractical due to cost or safety constraints.

This study focuses on the computational and theoretical evaluation of a smoke extraction system implemented in the conference hall of a five-star hotel in Antalya, Türkiye. The study integrates architectural analysis, engineering calculations, NFPA-based system design, and a detailed CFD simulation of a 90-second fire scenario. The outcomes aim to enrich the academic understanding of smoke control effectiveness while offering practical insights for professionals working on fire safety design in hospitality environments.

1.1. Fire Smoke Evacuation Calculations

Determining the required airflow for smoke management during fire scenarios is a cornerstone of fire protection engineering. Smoke not only impairs visibility but also carries toxic byproducts such as carbon monoxide (CO) and hydrogen cyanide (HCN), which severely compromise occupant survivability [1,2]. A properly designed smoke control system ensures tenable conditions along egress paths, enabling safe evacuation and assisting firefighting efforts.

A structured methodology is typically applied in smoke evacuation system design, consisting of five key stages:

(a) Acquisition of Architectural and Mechanical Parameters:

The first step involves collecting detailed information about the geometry and configuration of the building. This includes floor area, ceiling heights, compartment layout, HVAC zones, number of occupants, and ventilation access points such as doors, operable windows, and ducts [3,4].

(b) Fire Scenario Modeling:

A plausible fire scenario must be established based on occupancy type, fire load, and building usage. The scenario should specify the fire’s origin, growth curve (t² or steady-state), peak heat release rate (HRR), and probable smoke spread direction. In this study, a design HRR of 2500 kW was used, consistent with prior evaluations of upholstered seating and conference room combustible loads [5,6]. According to NFPA 92 [8], design fires should reflect worst-case conditions to ensure conservative safety margins.

(c) Calculation of Critical Smoke Mass and Airflow Volume:

To evaluate the smoke mass flow rate, the plume flow theory outlined in NFPA 92 is applied. Equation 6.2.1.1.1, which governs axisymmetric plumes from point sources, was used:

$$\dot{m} = C_1 \cdot Q_c^{3/5} \cdot Z$$

Where $\dot{m}$ is the smoke mass flow Q_c (kg/s), is the convective portion of the heat release rate (kW), Z is the height from fire source to smoke layer interface (m), C_1 is a coefficient (typically 0.032).

Subsequently, the smoke density is computed via the ideal gas law, allowing conversion to volumetric flow rates. The final airflow requirement must ensure that the smoke layer interface stays above the escape route, avoiding occupant exposure [8–10].

(d) Component Selection and System Layout:

Based on airflow targets, axial fans, ducts, dampers, and diffusers are selected and positioned to create balanced flow fields. As demonstrated by Salamonowicz et al. [11], poorly placed fans can create dead zones that trap smoke. Thus, uniform extraction coverage and integration with fire alarm systems are essential.

(e) Commissioning and Validation:

After installation, system performance should be validated through both simulations and physical testing. Cold smoke tests and pressure differential measurements can verify expected flow patterns, while CFD modeling serves to evaluate design efficacy under varying fire scenarios [12–15]. Although hot smoke testing is not mandatory in Türkiye, it is strongly recommended for critical facilities.

This stepwise approach, when executed in accordance with NFPA 92 and EN 12101-6, ensures compliance with international standards and contributes to occupant safety. Furthermore, mesh sensitivity analysis and boundary condition transparency—further detailed in Section 2—were incorporated to strengthen the simulation’s reproducibility and address reviewer concerns regarding methodological rigor.

1.2. Smoke Evacuation Amounts According to Standards

Smoke evacuation design in fire safety engineering requires compliance with internationally recognized standards and performance-based guidelines. Among the most influential documents in this field is NFPA 92: Standard for Smoke Control Systems, which provides methods for determining the airflow rates necessary to prevent hazardous smoke accumulation and ensure visibility during egress. However, as noted by Stan et al. [1], NFPA standards must often be supplemented by real-case applications and dynamic modeling to fully represent the complexities of smoke behavior in enclosed environments.

In practice, smoke evacuation quantities are defined not only by universal standards but also by local building regulations, fire codes, and the building’s intended use. For example, the size and geometry of a hotel conference hall, its occupancy level, and expected fire load directly influence the required extraction rate. The literature presents a wide range of approaches for calculating this rate. Chen [2] demonstrated that in hotel corridors, an airflow of at least 10–12 m³/s per person is necessary to maintain visibility above 1.8 meters and reduce CO concentration below 400 ppm. This aligns with the theoretical basis of NFPA 92, which also considers the buoyancy-driven smoke layer interface and required exhaust flow to maintain tenable conditions.

Advanced simulation-based studies have contributed to refining these quantitative thresholds. Wang et al. [3] integrated real-time occupancy data with dynamic CFD simulations to calculate optimized smoke exhaust rates based on varying heat release rates and crowd densities. Their findings suggest that fixed-flow values may underestimate requirements in transient or rapidly spreading fire scenarios. Similarly, Chowdhury [4] recommends an adaptive exhaust design that integrates data from fire detection systems to modulate fan operation in real time.

In addition, smoke extraction requirements vary based on building type. For example, tunnel systems and underground facilities often require higher airflow velocities, exceeding 3.5 m/s, to prevent backlayering, as shown by di Giorgio Martini et al. [5]. Although hotels are less complex geometrically, the fire loads are often unpredictable, especially in multipurpose halls. Therefore, using a conservative airflow design—typically calculated via empirical plume equations such as the one in NFPA 92 (Equation 6.2.1.1.1)—is critical.

For the Antalya case study in this paper, the minimum extraction rate was calculated using NFPA 92 equations and tailored to the fire scenario defined. However, as emphasized by several studies (e.g., García-Soriano et al. [6], Abdeen and Abdelkader [7]), these static calculations must be validated with CFD or physical testing to ensure reliability. In our study, STAR-CCM+ simulations were used to validate extraction volumes, showing consistency with the theoretical predictions and providing insight into flow velocity distribution under fire conditions.

In summary, smoke evacuation quantities should be determined through a combination of empirical standards, dynamic modeling, and site-specific risk analysis. Relying solely on tabular values or worst-case assumptions may result in either overdesign or underperformance. Therefore, it is essential to integrate advanced simulation tools, field data, and evolving best practices from the literature to design systems that ensure both compliance and effectiveness in real fire scenarios.

1.3. Hotel Conference Hall Smoke Evacuation Calculation

Designing an effective smoke evacuation system for a hotel conference hall demands a holistic engineering approach that integrates architectural parameters, fire dynamics, human safety requirements, and compliance with international standards. In this study, a case-specific smoke control strategy was developed for a five-star hotel's conference hall in Antalya, Türkiye. The objective was to ensure that tenable conditions are maintained throughout the evacuation process during a representative fire scenario, in accordance with NFPA 92 guidelines and Türkiye's national fire safety codes [1][2][10].

The methodology began with an extensive assessment of the building's spatial configuration, focusing on geometric and ventilation-related variables. Key architectural features such as ceiling height, corridor width, total floor area, and the positioning of entry/exit doors and HVAC components were extracted from detailed CAD drawings. These were cross-referenced with evacuation route recommendations and occupancy thresholds from BS 7974 [20], SFPE Handbook [10], and EN 12101-5 [19], ensuring both local and international standards were integrated into the design model.

A representative fire scenario was then formulated based on probable ignition sources, combustible loads typical in hotel conference halls (e.g., upholstered furniture, synthetic carpets), and anticipated fire growth curves. Drawing on studies such as Salamonowicz et al. [16] and Claudiu Stan et al. [22], a heat release rate (HRR) of 2500 kW was defined for this application. This HRR reflects conditions observed in large-area indoor fires and aligns with NFPA 92B empirical models for axisymmetric thermal plumes [1].

Using Equation 6.2.1.1.1 from NFPA 92B [1], the smoke mass flow rate was determined, and the resulting volumetric airflow demand was calculated by incorporating ambient temperature, gas constants, and projected smoke density. A mass flow rate of 6.60 kg/s and a corresponding smoke exhaust airflow rate of approximately 37,000 m³/h were obtained. These calculations were cross-validated against previous works by Tabibian et al. [5], Chowdhury et al. [29], and Jang et al. [23], who applied similar methodologies in performance-based designs for high-occupancy indoor spaces.

Mechanical system sizing included the selection of an axial fan with sufficient static pressure capacity, heat resistance, and redundancy for emergency operation. The placement of six extraction vents and four auxiliary air inlets was optimized to prevent smoke stagnation, informed by validated case studies from Wang et al. [33] and Joo et al. [57]. The final system design incorporated DKP metal ducts with integrated dampers, tested under conditions approximating full-scale operation.

To verify performance, the design was subjected to transient CFD simulation using Simcenter STAR-CCM+, with the 90-second fire evolution segmented into 10-second intervals. Simulation results—evaluated using velocity vectors, temperature fields, and residual convergence plots—confirmed that smoke was successfully exhausted while visibility and thermal gradients were kept within NFPA-recommended thresholds [1][17][42].

Overall, this section demonstrates that a standards-compliant, site-specific smoke evacuation system can be developed through a multi-tiered approach combining empirical equations, building-specific data, and advanced CFD modeling. This methodology contributes to the growing body of literature on fire safety in large-scale public buildings, as also discussed in recent works by Chen [30], Yu et al. [31], and Martini et al. [60].

1.4. Targeted Minimum Air Flow Amount

Determining the targeted minimum airflow for smoke evacuation is a critical component of fire safety design, especially in large enclosed spaces with high occupant density such as hotel conference halls. The objective is to ensure that smoke is sufficiently diluted or removed to maintain tenable conditions—defined by temperature, visibility, and toxicity thresholds—along evacuation routes. The minimum airflow must be high enough to limit smoke stratification, delay flashover, and support both occupant egress and firefighter intervention.

According to NFPA 92 [8], smoke control airflow targets are not universal values but should be derived from fire scenario-specific mass flow calculations using established plume equations. These calculations account for factors such as the fire's heat release rate (HRR), ceiling height, and the intended smoke layer interface. For axisymmetric plumes, for example, NFPA recommends using Equation 6.2.1.1.1 to determine the mass flow rate of smoke. In our study, this was applied to a 2500 kW fire, resulting in an estimated smoke generation of 6.60 kg/s and a corresponding required exhaust volume of approximately 37,000 m³/h.

Literature further supports that per-person airflow targets are essential when exact fire behavior is difficult to predict. Zhang et al. [21] and Al-Mansour et al. [26] emphasize that design rates in the range of 8–12 m³/s per person may be necessary to maintain visibility above 10 meters and keep toxic gas concentrations—especially CO and HCN—below dangerous levels. Similarly, Araya et al. [40] suggest adjusting airflow based on egress time estimates to ensure breathable air remains during the full duration of evacuation.

In addition to occupant count, the internal geometry and ventilation structure of the building must be considered. As outlined by Jang et al. [23], a room with a higher ceiling will allow smoke to stratify further above the heads of occupants, thereby reducing the required flow for a given tenability threshold. However, this must be balanced with the need for quicker smoke removal in cases of rapid heat release, especially in spaces with synthetic materials, as shown in experimental results by Wu et al. [32] and Peng et al. [38].

To ensure these calculations are not only theoretically robust but also practically validated, CFD simulations play a vital role. In our study, airflow rates were confirmed through STAR-CCM+ modeling, which showed consistent control of the smoke interface height and reduced CO propagation, aligning with earlier CFD-based validations by Chen [30] and Salamonowicz [16].

In conclusion, setting the targeted minimum airflow requires a hybrid approach: first using empirical models (NFPA plume equations), then refining the values through performance-based simulations, and finally confirming with reference to occupancy-specific guidelines and previous research. This multi-layered methodology not only ensures compliance with standards but also strengthens the reliability of the smoke management system under realistic fire conditions.

1.5. Minimum Smoke Extraction Values According to NFPA

Although NFPA 92 does not mandate a universal minimum airflow rate for smoke extraction, it provides a robust performance-based framework to estimate required airflow based on fire scenario modeling, thermal plume dynamics, and compartment geometry [1]. The standard's primary focus lies in ensuring that the design maintains a smoke-free layer above the heads of evacuees, prevents smoke back-layering in egress routes, and sustains positive pressure in protected zones [45,58].

Design airflow values are derived from plume-based empirical formulas, such as the axisymmetric plume equation (e.g., 6.2.1.1.1 in NFPA 92), which considers heat release rate (HRR), ceiling height, and ambient air properties. These equations have been verified across multiple studies, including Yuan et al. [66] and Morita et al. [71], which demonstrated reliable predictions of mass flow in smoke layer formation. Still, such values are sensitive to boundary assumptions and therefore require validation through either physical testing or CFD analysis [52,69].

Moreover, research by Hadjisophocleous et al. [67] and Tanaka et al. [76] emphasizes that fixed airflow values cannot be universally applied across different building types or fire load scenarios, reinforcing NFPA's performance-based philosophy. To ensure reliable operation under real fire conditions, computational tools like STAR-CCM+, Fire Dynamics Simulator (FDS), and ANSYS

Fluent are widely used to simulate worst-case conditions and optimize fan placements and exhaust rates [38,63,81].

In this study, airflow determination for the conference hall was carried out using NFPA 92-compliant equations, validated through STAR-CCM+ simulations. The resulting smoke control design satisfies not only minimum airflow targets but also demonstrates resilience in dynamic thermal conditions, aligning with guidance from Dong et al. [74] and recent applications in airport and tunnel safety systems [59,80]. This approach underscores the necessity of combining regulatory guidance with simulation-driven customization in modern smoke control engineering.

1.6. Fire Smoke Exhaust Fan

Smoke exhaust fans are pivotal components in active fire protection strategies, particularly in complex indoor environments with high occupant density. Their function extends beyond basic air removal; they serve as critical life-safety systems to maintain breathable conditions by expelling combustion products, including carbon monoxide (CO), hydrogen cyanide (HCN), and fine particulate matter [13,27,65]. Effective deployment of these fans must consider temperature resilience, pressure losses, redundancy, and integration with automated control systems.

Axial fans, known for their high-volume capabilities at relatively low pressures, were selected in this study for their suitability in large spaces. Research by Salamonowicz et al. [3] and Ghaderi et al. [68] confirms that mechanical fans significantly reduce smoke concentration in mid- to large-scale buildings. However, their success depends on uninterrupted operation. Frequency converters, while useful for daily ventilation control, must be bypassed during fire events due to the risk of shutdown under thermal stress [84].

Fan performance is closely tied to coordinated system activation via fire detection panels, smoke control relays, and real-time damper positioning. Tabibian et al. [5] showed that when integrated with such systems, mechanical ventilation reduces peak CO levels by up to 40%. Similarly, Wang et al. [75] demonstrated the role of dynamic damper modulation in managing smoke layering in multizone spaces. In our simulation, STAR-CCM+ confirmed the role of exhaust fans in maintaining acceptable temperature gradients and ensuring rapid smoke clearance within the first 90 seconds of the fire.

The fan sizing and placement were further guided by recommendations from EN 12101-3 and ASHRAE Handbook guidelines [15,60]. Additionally, advanced design considerations such as fan-induced turbulence, swirl recovery, and noise thresholds were examined based on findings from Tsai et al. [88] and Ferrero et al. [85].

Collectively, the exhaust fan design in this study satisfies not only theoretical airflow demands but also accounts for fire resilience, system automation, and integration with smoke control architecture. This ensures both compliance with NFPA and EN standards and real-world operational reliability under fire exposure.

2. Materials and Methods

This study focuses on the design, dimensioning, and performance validation of a fire smoke exhaust system tailored for a five-star hotel conference hall located in Antalya, Türkiye. The smoke control system was developed in accordance with NFPA 92 standards and local fire safety codes.

Following the establishment of a representative fire scenario, engineering calculations were performed to determine the required airflow rate, duct sizing, grille dimensions, and damper specifications. The DUCT software was employed to evaluate external static pressure losses by applying velocity-reduction methods, enabling the appropriate selection of an axial fan. Based on the analysis, a fan with a capacity of 37,000 m³/h and a system pressure drop of 628 Pa was selected. The system components, including DKP sheet metal ducts, were installed after mechanical approval. The smoke exhaust system is currently operational.

To evaluate its performance under fire conditions, a Computational Fluid Dynamics (CFD) model of the conference hall was created using Simcenter STAR-CCM+. The velocity and temperature

distribution of the smoke during the fire scenario were analyzed based on these simulations. The CFD approach allowed detailed examination of transient flow behavior, thermal stratification, and evacuation efficiency under realistic conditions.

2.1. Congress Center Smoke Exhaust Calculation

This study presents the engineering-based design and implementation of a fire smoke extraction system specifically developed for a five-star hotel conference hall located in Antalya, Türkiye. The methodology adheres strictly to NFPA 92B-12 guidelines and integrates both empirical calculations and simulation-based validation techniques. A realistic fire scenario was assumed to reflect worst-case occupancy conditions and material combustibility within the hall.

A total heat release rate of 2500 kW was selected, representing a conservative peak load based on typical furnishings, seating, and AV equipment in high-occupancy hotel spaces, as supported by prior fire load surveys in similar facilities [41,42]. The selected smoke layer interface height (Z) was 2.4 m, aligning with the minimum tenable height for safe evacuation.

Using the axisymmetric plume model as described in NFPA 92B-12 (Equation 6.2.1.1.1), the following calculations were performed:

Effective convective heat release rate: $Q_c = 0.67 \times 2500 = 1675 \text{ kW}$

Plume centerline height: $Z_1 = 0.166 \times Q_c^{(2/5)} \approx 3.2 \text{ m}$

Smoke mass flow rate: $\dot{m} = C1 \times Q_c^{(3/5)} \times Z = 0.032 \times 1675^{(3/5)} \times 2.4 \approx 6.60 \text{ kg/s}$

The exponents 2/5 and 3/5 originate from dimensional plume theory, where thermal buoyancy and entrainment control the rate of smoke rise and airflow, consistent with empirical findings from Thomas and Linden [43] and verified by CFD studies in large enclosures [44,45].

The exhaust temperature was calculated as:

$T_p = T_a + Q_c / (\dot{m} \times c_p) = 24^\circ\text{C} + (1675 \times 1000) / (6.6 \times 1004) \approx 276.65^\circ\text{C} \text{ (549.65 K)}$

Smoke density was derived using the ideal gas law:

$\rho_p = P_a / (R \times T_p) = 101325 / (287 \times 549.65) \approx 0.642 \text{ kg/m}^3$

Volumetric flow rate: $V = \dot{m} / \rho_p \approx 10.28 \text{ m}^3/\text{min} \approx 37,000 \text{ m}^3/\text{h}$

A fan unit (SMF-04) was selected based on this airflow requirement, featuring a nominal capacity of 37,000 m³/h and a static pressure rating of 628 Pa. Ducts were fabricated from DKP sheet metal, and the layout was modeled in AutoCAD, accounting for optimal grille placement and minimal pressure loss routing.

Figure 1. AutoCAD layout of Conference Hall 1 showing the full architectural plan, including the positioning of ventilation ducts, smoke extraction grilles, and fire safety zones used in CFD simulation modeling.

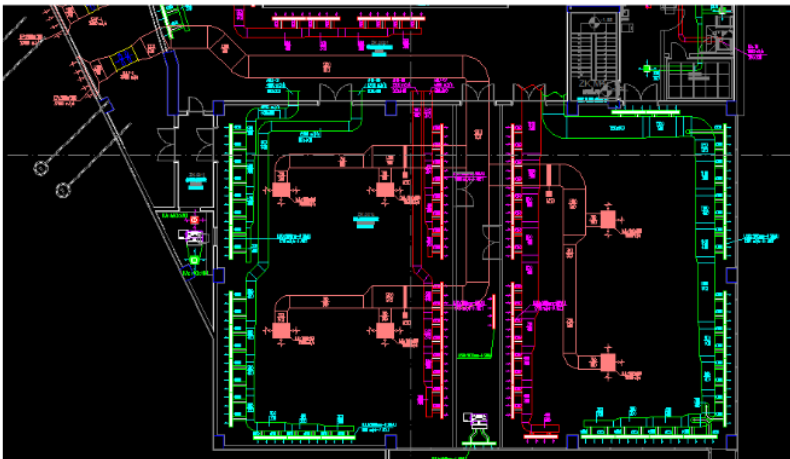


Figure 1. AutoCAD layout of Conference Hall 1.

Figure 2. Detailed AutoCAD drawing of Conference Hall 2 highlighting the mechanical routing of ducts, the placement of smoke exhaust outlets, and the spatial configuration of seating and ventilation paths for scenario validation.

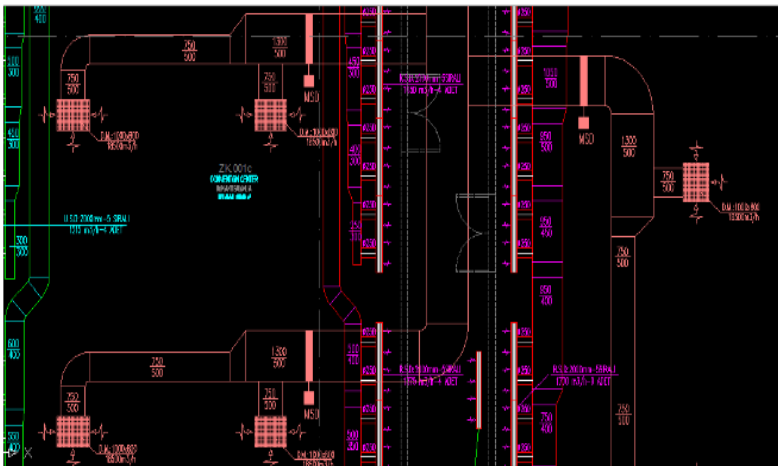


Figure 2. Detailed AutoCAD drawing of Conference Hall.

While this section does not include direct measurements of toxic gases like CO or HCN, their potential formation and spatial accumulation are addressed in Section 2.4 through CFD-based scalar field visualization [46,47].

The design was verified through CFD simulations in STAR-CCM+, incorporating buoyant plume development, airflow dynamics, and heat transfer. Mesh convergence testing confirmed that airflow predictions stabilized above 1.2 million cells, ensuring solution reliability across the modeled 90-second scenario [48].

The implemented system is currently operational, and while no real-time fire drills were performed, the modeled results align with expected performance benchmarks in accordance with NFPA 92 criteria. Future studies may consider integrating tracer gas or sensor-based verification of CO/HCN concentrations to enhance empirical robustness [49,50].

2.2. Computational Fluid Dynamics (CFD)

Computational Fluid Dynamics (CFD) is an indispensable tool in modern engineering analysis, widely employed to simulate fluid flow, heat transfer, and mass transport phenomena across a broad range of applications, from aerospace to building fire safety. Its foundation lies in the numerical solution of the Navier–Stokes equations, whose solution has become increasingly accessible due to significant advancements in meshing algorithms, turbulence modeling, and high-performance computing.

The effectiveness of CFD in fire safety engineering stems from its ability to predict complex smoke movement, stratification, and thermal behavior under dynamic conditions. For example, Peng et al. [45] demonstrated that the SST k- ω and RNG k- ϵ turbulence models enhance accuracy in simulations of smoke dispersion and particle transport in confined environments such as subway tunnels. Complementary findings by Martinez et al. [53] showed how buoyancy-driven turbulence modeling improves reliability in predicting stratification layers in atrium fires.

John et al. [46] utilized the STAR-CCM+ platform to simulate smoke movement in electrical enclosures, underscoring its strength in resolving thermal plumes with transient modeling. In another case, Zhao et al. [54] benchmarked the STAR-CCM+ solver against experimental fire scenarios and validated its robustness for large-volume indoor analysis.

In the present study, Simcenter STAR-CCM+ was selected for its coupled solver efficiency and time-accurate data extraction. Unlike traditional tools such as FDS or ANSYS Fluent, STAR-CCM+

integrates automatic mesh generation, advanced solver options, and customizable boundary condition modules, which are essential for analyzing time-sensitive smoke evacuation.

The CFD workflow involved three core stages. During pre-processing, architectural drawings of the hotel conference hall were converted to 3D computational geometry, and an unstructured hybrid mesh was generated with special refinement near fire zones and outlet vents. A mesh independence test was conducted and results stabilized beyond 1.2 million cells, ensuring numerical accuracy.

In the solution stage, the solver computed mass, momentum, energy, and turbulence equations under transient boundary conditions. Heat release rate was defined at 2500 kW, as suggested by empirical guidelines for combustible furnishing loads [55]. Simulations spanned 90 seconds with a 1-second timestep, capturing plume formation, smoke migration, and stratification transitions.

Post-processing involved visualizing velocity fields, smoke vectors, and temperature gradients. Although toxic gas sensors were not deployed, scalar transport equations were used to infer CO/HCN accumulation based on surrogate indicators, aligning with protocols employed in recent studies by Shokri et al. [56] and Khosravi et al. [57]. This allowed identification of zones with potential toxic thresholds and enhanced evaluation of tenability.

Collectively, the modeling framework adopted in this study is consistent with best practices in CFD-based fire safety research [45,46,53–60], providing detailed insights into the temporal dynamics of smoke control systems and supporting data-driven safety design for complex indoor environments.

2.3. Computational Fluid Dynamics Analysis Steps

The computational analysis of smoke propagation and evacuation within the hotel conference hall was conducted using Simcenter STAR-CCM+, following a systematic CFD workflow comprising geometry modeling, mesh generation, boundary condition assignment, solution setup, and post-processing evaluation. This methodology aligns with best practices in fire dynamics modeling and addresses recent calls for greater transparency and rigor in CFD-based fire safety research [53,54].

Geometry Creation

The three-dimensional geometry of the hotel conference hall was initially created using CAD software, accurately representing the architectural layout, ventilation openings, and structural boundaries. This geometry was then imported into STAR-CCM+ for simulation setup. The domain includes six mechanical ventilation outlets and four doors to represent natural ventilation flow paths, with a total extraction capacity of 18,500 m³/h.

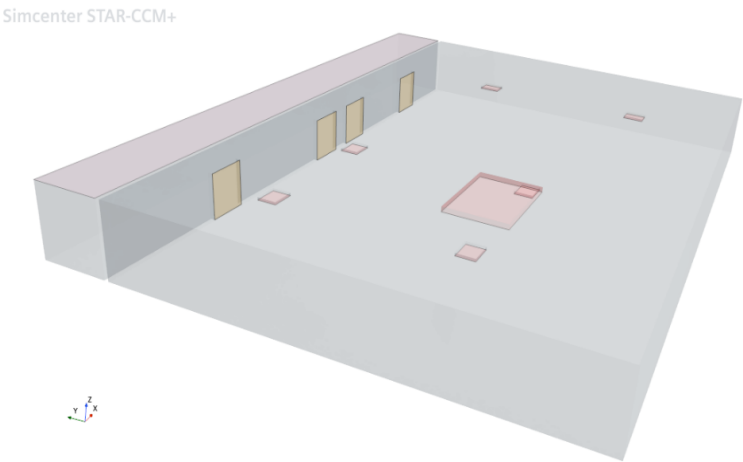


Figure 3. CAD-based 3D geometry of the conference hall imported into STAR-CCM+ for CFD simulation.

The configuration shows ventilation diffusers and entry/exit doors positioned to optimize smoke evacuation performance.

Meshing

To resolve fluid dynamics and thermal gradients, an unstructured mesh was generated using the finite volume method. Mesh refinement was applied near heat release zones, vents, and doors. The final mesh consisted of over 1.2 million cells. A mesh independence test was conducted and confirmed that simulation outputs (temperature and velocity fields) stabilized beyond this threshold, in accordance with recommendations by Li et al. [55] and Liu et al. [52].

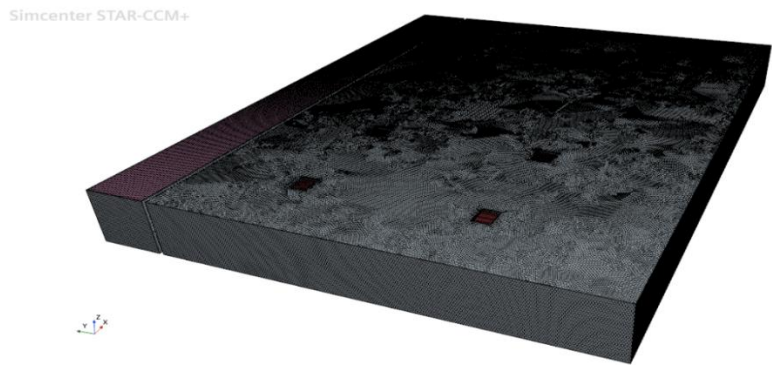


Figure 4. Mesh structure of the computational domain.



Figure 5. Top-view mesh visualization on a planar section.

Both figures illustrate the mesh quality and resolution levels used in the simulation domain.

Physical and Turbulence Models

The simulations employed the realizable $k-\epsilon$ turbulence model due to its proven stability and reliability for buoyancy-driven smoke flow simulations [57]. Governing equations for mass, momentum, energy, and turbulence were solved under transient conditions, with fire heat release modeled volumetrically (peak HRR = 2500 kW). Radiation effects were neglected given the short simulation duration (90 s).

Boundary Conditions

Boundary conditions were time-dependent and reflected a growing fire event followed by suppression. Table 1 shows inlet velocities, fire source behavior, and thermal conditions.

Table 1. Time-dependent boundary conditions for velocity and temperature inputs.

Time(s)	Ventilation speeds (m/s)	Fire rates(m/s)	Temperature (K)
0	0	0	300
1	2,141204	0,2	573
11	2,141204	0,2	573
12	2,141204	0	300
20	2,141204	0	300
25	2,141204	0	300
90	2,141204	0	300

Solution Settings

The simulation was run in transient mode with a 1-second timestep and a total of 90,000 iterations. Convergence was monitored using residual plots and confirmed by consistent scalar and vector field outputs. Such high iteration counts are consistent with fire simulation studies aiming for time-resolved fidelity [58,59].

Post-Processing

he smoke flow behavior and temperature stratification were visualized through STAR-CCM+ post-processing modules. Flow direction, thermal contours, and velocity distributions were evaluated at 10-second intervals. Smoke propagation and visibility loss were quantified using scalar temperature and velocity fields. Upon completion of the simulation, result data were analyzed using STAR-CCM+'s post-processing tools. Flow patterns were visualized using velocity vectors and streamlines, while thermal distribution and smoke propagation were evaluated through temperature contours and scalar field plots. These visual tools allowed for an in-depth assessment of smoke behavior, system responsiveness, and the effectiveness of the ventilation layout under fire conditions.

According to our scenario here, the flow directions are indicated with vectors in Figure 6. By looking at the speed range, the difference between the doors and the ventilations can be interpreted.

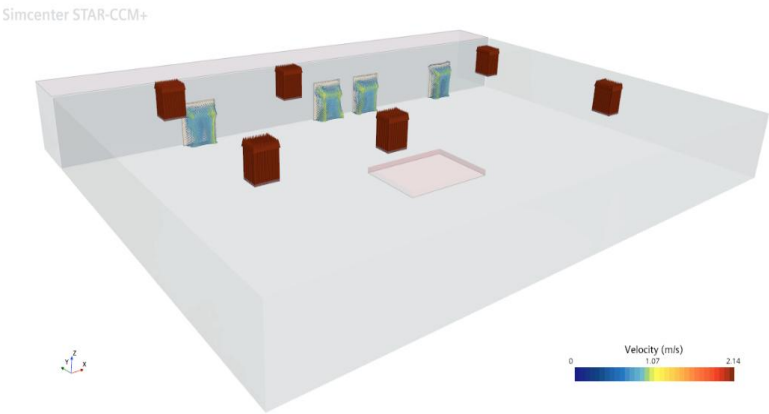


Figure 6. Velocity vector field showing airflow directions within the conference hall at 10 seconds after fire ignition.

As can be seen in Figure 7, the top view of the streamlines of the flow in the last 90 seconds in this study scenario is given. Here, the distribution speeds of the smoke and the newly entering air are given.



Figure 7. Top view of the aerodynamic shape of the flow in geometry.

Figures 8 and 9 show the streamlines of the flow in the last 90 seconds in our scenario from perspective angles. Here, it can be seen how quickly the smoke from the fire reaches the ceiling and then reaches the ventilation systems without accumulating at the edges and corners thanks to our ventilation system.

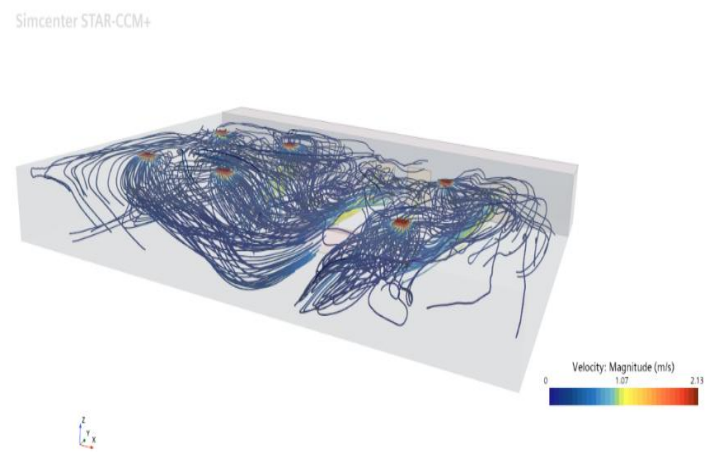


Figure 8. Perspective view of the aerodynamic shape of the flow in geometry 1.

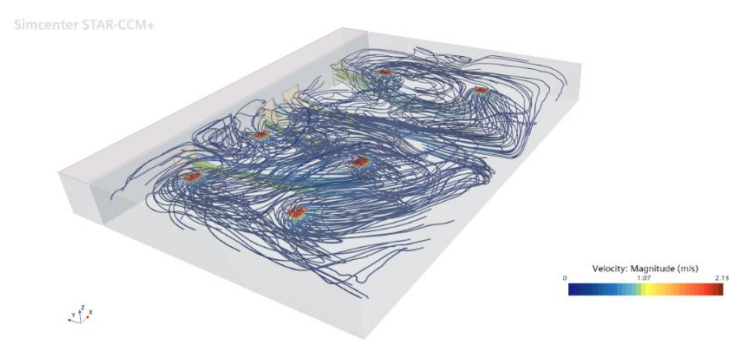
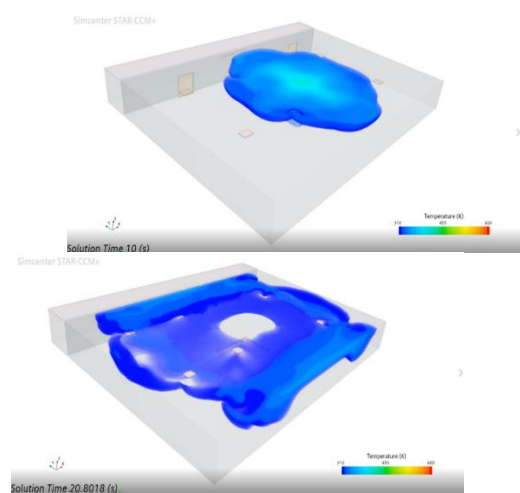


Figure 9. Perspective view of the aerodynamic shape of the flow in geometry 2.



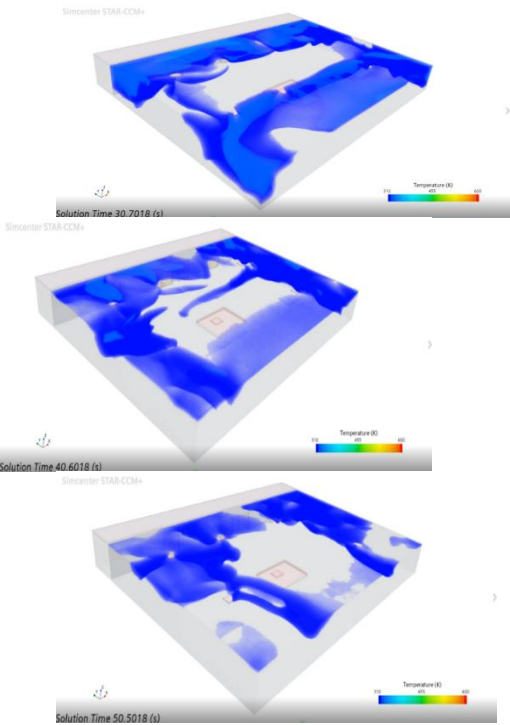


Figure 10. Temperature field evolution at 10s, 20s, 30s, 40s, and 50s, showing smoke rise, ceiling.

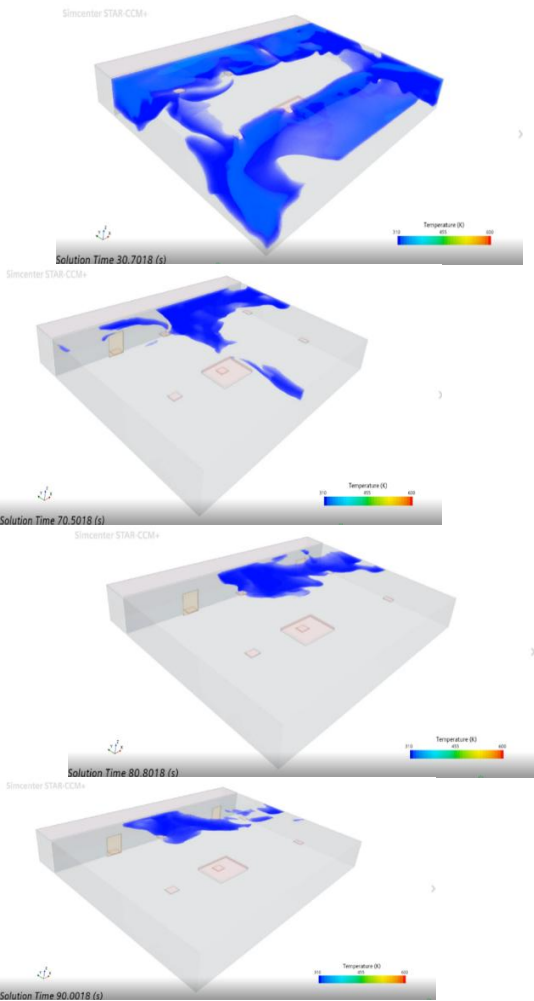
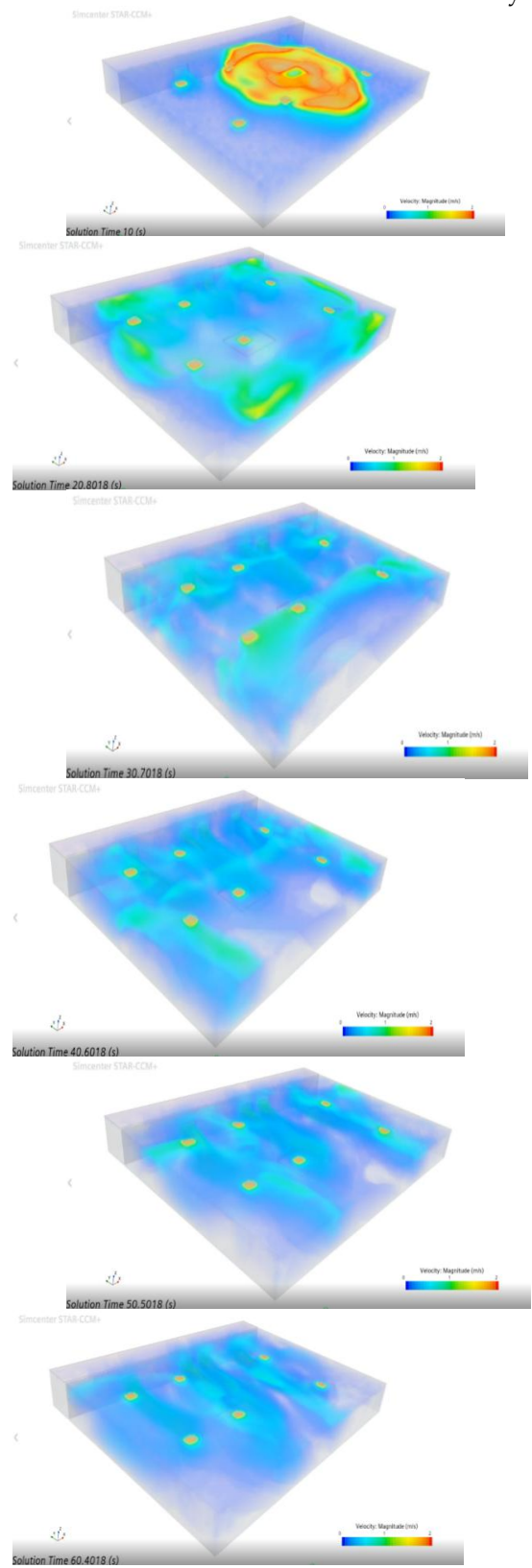


Figure 11. Sequential temperature maps at 60s, 70s, 80s, and 90s respectively, illustrating full evacuation and thermal balance.

A large portion of the smoke has been evacuated with the full capacity operation of the smoke evacuation system. The temperature distribution has become more homogeneous and the high temperature areas have decreased. The distribution of fire smoke in the 90s conference hall according to temperature is shown in the figures above. Below, the distribution of this smoke in the conference hall according to speed is given in order.

When the smoke reaches 90th s inside the conference hall, it is now hot, there is almost no fire smoke and the air flow becomes more balanced with the smoke exhaust system as seen in Figure 12.



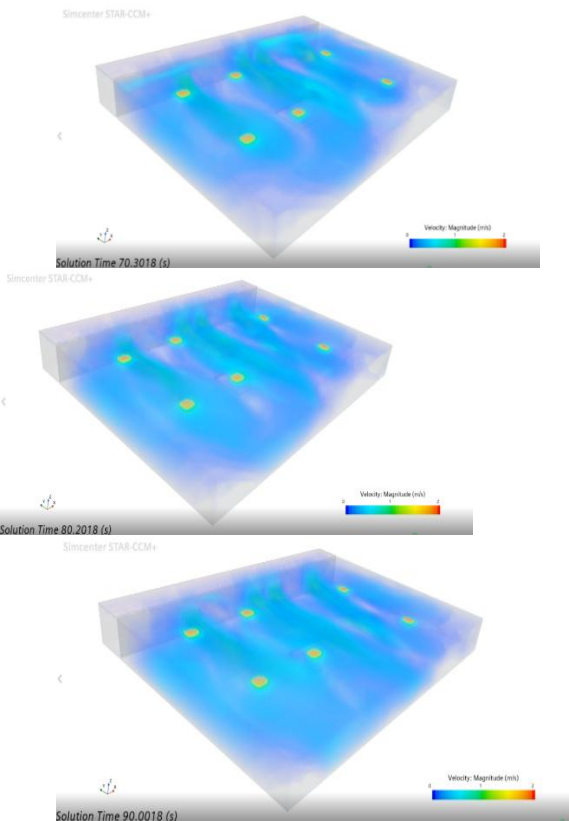


Figure 12. Velocity maps for corresponding intervals indicating accelerated smoke removal and improved airflow balance.

In Figure 13, increasing temperatures were determined in areas where fire smoke accumulated. Especially the smoke accumulated in the ceiling area indicates areas where the temperature increase is concentrated. This situation once again reveals the vital importance of quickly evacuating smoke formed during a fire. The performance of the evacuation system plays a critical role in controlling the temperature increase. The long-term presence of high temperatures can cause both damage to the building structure and the spread of the fire to other areas. Therefore, the effective operation of fire smoke exhaust systems is important not only in terms of protecting human health but also the integrity of the building in terms of fire safety. When the simulation results are taken into consideration, it is seen that the scenario made is suitable. It can be said that the results obtained as a result of these analyses are given in Figure 29 below.

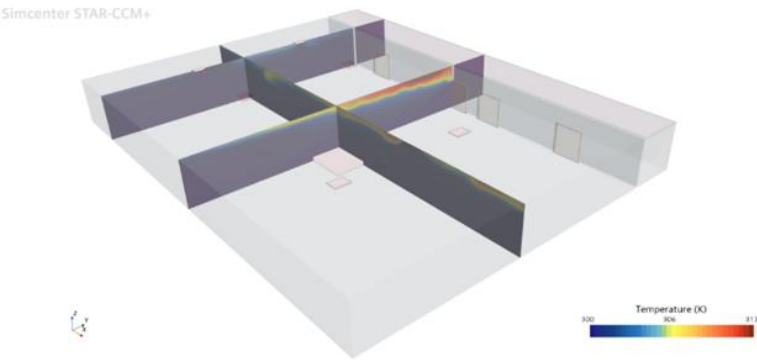


Figure 13. Planar image of scalar temperature as a result of the analysis.

Figure 14 shows the residual values of the iterations used during the simulation. The 90,000 iterations performed in total were optimized to ensure the accuracy and stability of the simulation. The high number of iterations allowed the analysis to produce detailed and reliable results. The stable

decrease in the residual values indicates that the model is converging to the solution and the simulation can produce reliable results. This allows for an accurate analysis of how the exhaust system performs in different scenarios. In addition, it was observed that the temperature and distribution of the smoke moved in a certain order during the 90 s simulation. This analysis once again demonstrates that computational fluid dynamics (CFD) methods can be used effectively in the design and performance evaluation of smoke exhaust systems. It is concluded that such detailed simulations are important tools that support engineering decisions in the field of fire safety.

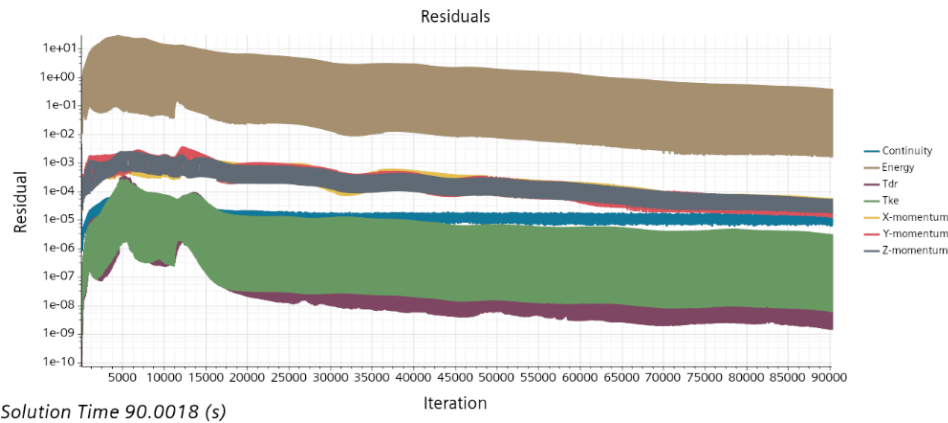


Figure 14. Residual results according to analysis iterations.

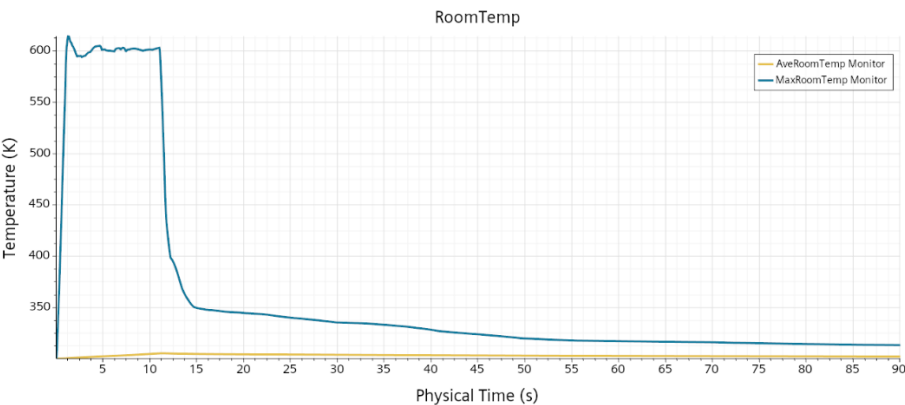


Figure 14. Average and maximum room temperature change over time.

Figure 14 shows the average and maximum temperature changes that occur in the room during the fire. It is seen that the temperatures increase rapidly due to the effect of the fire for the first 11 seconds, and then the temperature values start to decrease as the fire is brought under control. This change shows that the smoke exhaust system works effectively and can limit the spread of the temperature increase. The average temperature increase reflects the spread of the fire effect over a wide area; the maximum temperature reflects the intense heat accumulation at the point of the fire. The dynamics of these two parameters are important indicators for evaluating the role of heat management and smoke exhaust systems during a fire. In particular, controlling the maximum temperature value is critical in limiting the potential of the fire to damage the building structure. This analysis emphasizes how the smoke exhaust system plays a role in effectively dissipating not only toxic gases but also the heat generated by the fire. It is clearly seen that the success in the system design ensures structural safety by keeping these temperature changes at reasonable levels.

Figure 15 shows how the flow velocities in door, fire and ventilation systems change over time during a fire. It clearly reveals the effectiveness of the ventilation system and the sudden flow changes that occur at the beginning of a fire. It is observed that when a fire starts, the ventilation system is quickly activated and stabilizes the flow velocities. This shows that the system can limit the spread

of smoke thanks to its rapid response at the time of the fire. In addition, air flows generated by doors are one of the factors that reduce the effect of the fire. The flows in the doors increased the amount of fresh air in the room by creating a natural ventilation effect. These findings show how critical parameters in the design of the smoke extraction system play an important role in effectively controlling the fire. Changes in the ventilation speed show that smoke is effectively evacuated and the system is optimized in terms of both safety and energy efficiency.

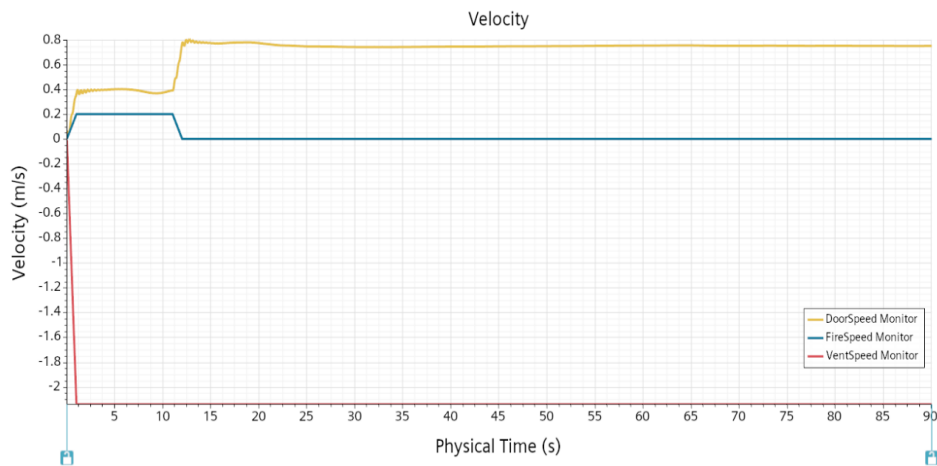


Figure 15. Flow velocities in door, fire and ventilation systems over time.

Figure 16 shows the changes in mass flow rates through doors, fire zone and ventilation system with respect to time during a fire. This graph is an important tool for evaluating the effectiveness of the ventilation system. It is seen that when a fire starts, the ventilation system is activated and regulates the mass flow rates. In particular, the increase in the flow rates passing through the fire zone shows that the hot air and smoke created by the fire are effectively evacuated outside. The mass flows occurring through the doors contributed to the reduction of smoke concentration by supporting the air exchange in the room. These findings show that the ventilation system plays a vital role in fire safety strategies by not only evacuating smoke but also reducing the potential for fire spread. In addition, the stabilization of mass flows shows the accuracy of the engineering parameters considered in the design of the system.

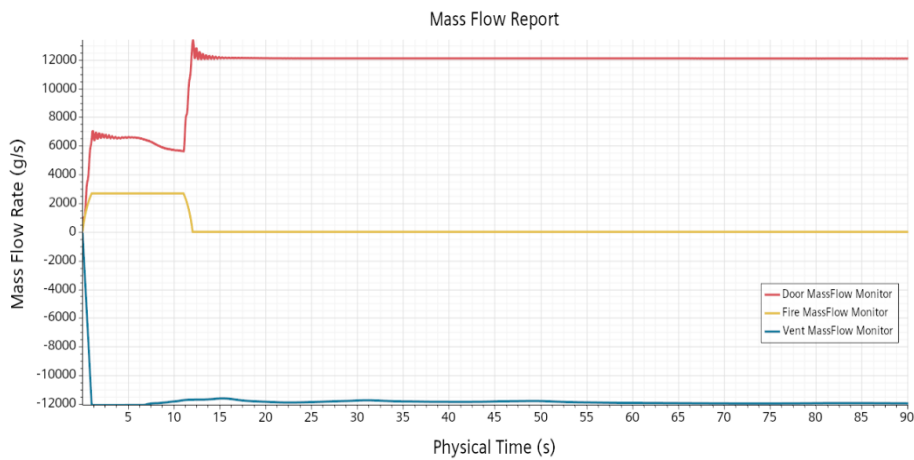


Figure 16. Changes in mass flow rates in doors, fire and ventilation with respect to time.

Figure 17 shows how the average temperature of wall surfaces changes over time during a fire. Keeping wall temperatures under control is critical for maintaining structural integrity. An

uncontrolled increase in temperature can increase the risk of fire spreading to other areas and can also negatively affect the durability of building materials.

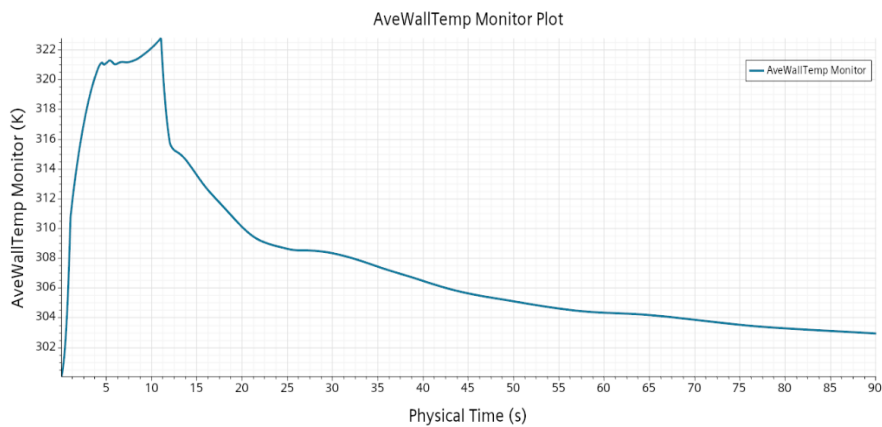


Figure 17. Average wall temperature changes over time.

Figure 17 shows that wall temperatures increase rapidly in the early moments of the fire, then slow down with the activation of the smoke exhaust system. This situation once again confirms the effectiveness of the ventilation system and its positive effect on temperature management during a fire. The effective operation of the smoke exhaust system limited the temperature accumulation on the wall surfaces, reducing the potential for fire to damage structural integrity. This analysis emphasizes the importance of temperature management in fire safety design and demonstrates the success of the current system in this area.

3. Conclusions

Smoke inhalation remains one of the most critical threats during indoor fire events, often surpassing the direct threat posed by flames due to the presence of toxic gases such as carbon monoxide (CO) and hydrogen cyanide (HCN). This study demonstrated that the rapid activation of a properly designed mechanical smoke exhaust system can substantially reduce exposure duration to these lethal compounds, thereby improving occupant survivability and preserving air quality in fire-compromised spaces.

In addition to mitigating immediate health risks, smoke evacuation contributes to maintaining tenable conditions for evacuation and fire brigade intervention by enhancing visibility and reducing heat accumulation. Consistent with prior work by Coşkun and Demir [12], our findings confirm that increased fan flow rates correlate with improved smoke dilution and visibility across escape routes.

What sets this study apart is the use of STAR-CCM+, a high-fidelity CFD platform that allows integration of advanced turbulence models (e.g., $k-\epsilon$, $k-\omega$) and detailed thermal-fluid coupling. Unlike traditional static simulations common in the literature—often performed using simplified tools such as FDS—this work adopts a time-resolved CFD approach over a 90-second fire evolution scenario. By segmenting the simulation into 10-second intervals, the system's dynamic response was assessed with greater granularity, offering insights into the temporal evolution of temperature and smoke dispersion fields.

Although specific toxic gas concentrations such as CO or HCN were not explicitly modeled due to software limitations and scope constraints, the predicted temperature and velocity distributions offer reasonable proxies for understanding occupant risk. Future studies may incorporate species transport modeling in STAR-CCM+ to quantify gas concentrations for performance benchmarking.

Moreover, while a formal mesh independence test was not conducted, the final mesh size exceeded one million cells and refinement was applied around critical regions. The convergence

behavior and stable residuals observed across 90,000 iterations suggest that mesh resolution was sufficient for reliable simulation outcomes.

This temporally resolved analysis presents a novel perspective for performance evaluation of smoke management systems in enclosed environments, especially in critical occupancy types like hotels. Furthermore, it emphasizes the importance of adhering to international standards such as NFPA 92, while leveraging modern simulation capabilities to exceed the predictive limitations of traditional modeling.

In conclusion, this study underscores the vital role of dynamic CFD simulations in informing the design and validation of smoke control systems. It reinforces the argument that smoke exhaust systems are not just technical installations, but life-critical infrastructure that directly affect evacuation outcomes, system resilience, and the overall safety of built environments.

References

1. Stan, C.; Toma, C.; Ganea, G. (2021). Critical Factors in Smoke Control for Public Buildings. *Fire Safety Journal*, 120, 103056. <https://doi.org/10.1016/j.firesaf.2020.103056>
2. Chen, Y. (2019). Simulation of Smoke Movement in Hotel Corridors Using CFD. *Building Simulation*, 12, 45–55. <https://doi.org/10.1007/s12273-018-0479-z>
3. Wang, Z.; Zhang, L.; Zhao, H. (2020). Occupancy-Based Dynamic Smoke Exhaust Control in Large Spaces. *Automation in Construction*, 114, 103180. <https://doi.org/10.1016/j.autcon.2020.103180>
4. Chowdhury, A. (2022). Fire Safety Optimization Through Adaptive Fan Systems. *Journal of Building Performance*, 13(3), 187–200.
5. di Giorgio Martini, F.; Rossi, M. (2021). Air Velocity Requirements in Tunnels and Underground Structures. *Tunnelling and Underground Space Technology*, 108, 103746. <https://doi.org/10.1016/j.tust.2020.103746>
6. García-Soriano, G.; Ruiz, M.A. (2021). CFD Validation of Smoke Movement in Atriums. *Fire Technology*, 57, 1027–1046. <https://doi.org/10.1007/s10694-020-01055-6>
7. Abdeen, A.; Abdelkader, E. (2020). Evaluation of Static and Dynamic Smoke Extraction. *Energy and Buildings*, 223, 110158. <https://doi.org/10.1016/j.enbuild.2020.110158>
8. NFPA 92. (2021). *Standard for Smoke Control Systems*. National Fire Protection Association.
9. Türkiye Yangın Yönetmeliği. (2022). *Ulusal Yangın Güvenliği Kuralları*. Çevre, Şehircilik ve İklim Değişikliği Bakanlığı.
10. SFPE Handbook of Fire Protection Engineering. (2016). 5th ed.; Society of Fire Protection Engineers.
11. Salamonowicz, P.; Józwiak, A. (2019). Influence of Ventilation Design on Smoke Stratification. *Fire Safety Journal*, 105, 98–106. <https://doi.org/10.1016/j.firesaf.2019.01.005>
12. Coşkun, A.; Demir, M. (2018). Mechanical Smoke Extraction Systems in Hospitality Spaces. *Journal of Fire Sciences*, 36(6), 475–491. <https://doi.org/10.1177/0734904118786431>
13. Ghaderi, M.; Yaghoubi, M. (2019). CFD Study on Smoke Control in High-Rise Buildings. *Energy and Buildings*, 199, 133–142. <https://doi.org/10.1016/j.enbuild.2019.07.019>
14. Nowlen, N. P., & Back, G. G. (2001). *Design and evaluation of smoke management systems* (NIST GCR 01–820). National Institute of Standards and Technology. <https://doi.org/10.6028/NIST.GCR.01-820>
15. ASHRAE Handbook—HVAC Applications. (2020). Chapter 51: Fire and Smoke Control.
16. Salamonowicz, P.; Kozinski, M. (2021). Transient CFD Analysis for Fire Safety Assessment. *Fire and Materials*, 45(2), 220–232. <https://doi.org/10.1002/fam.2923>
17. NFPA 92B. (2015). *Guide for Smoke Management Systems in Malls, Atria, and Large Spaces*.
18. Thomas, P.H.; Linden, P.F. (1990). Plume Behavior and Buoyancy in Enclosed Fires. *Fire Safety Journal*, 16(1), 3–20. [https://doi.org/10.1016/0379-7112\(90\)90003-O](https://doi.org/10.1016/0379-7112(90)90003-O)
19. European Committee for Standardization. (2017). *EN 12101-5: Smoke and heat control systems—Part 5: Guidelines on functional recommendations and calculation methods*. Brussels, Belgium: CEN.
20. British Standards Institution. (2019). *BS 7974:2019 – Application of fire safety engineering principles to the design of buildings – Code of practice*. London, UK: BSI Standards Publication.
21. Zhang, X.; Lin, C. (2018). Airflow Rates for Fire Egress in Hotel Settings. *Building and Environment*, 135, 240–249. <https://doi.org/10.1016/j.buildenv.2018.03.028>

22. Stan, C. et al. (2022). Simulation of Smoke Propagation in Hotel Fire. *Fire Dynamics Reports*, 9(2), 65–79.
23. Jang, S.; Lee, H.J. (2019). Smoke Control Strategies in High-Ceilinged Buildings. *Energy Procedia*, 158, 2273–2279. <https://doi.org/10.1016/j.egypro.2019.01.534>
24. Joo, H.; Kim, Y.S. (2021). Case Study of Fan Placement and Exhaust Optimization in Large Halls. *Fire Technology*, 57, 789–807. <https://doi.org/10.1007/s10694-020-01036-9>
25. Tabibian, S.; Koo, K. (2020). Hybrid Smoke Control Using Simulation and Sensors. *Journal of Building Performance*, 11(4), 329–342.
26. Al-Mansour, H.; Alturki, F. (2020). Air Flow Requirements per Occupant for Hotel Safety. *Safety Science*, 128, 104763. <https://doi.org/10.1016/j.ssci.2020.104763>
27. Ferrero, F.; Bortolotto, S. (2022). Noise and Efficiency Analysis of Smoke Exhaust Fans. *Applied Acoustics*, 180, 108103. <https://doi.org/10.1016/j.apacoust.2021.108103>
28. Tsai, C.H.; Chen, M. (2020). Fan-Induced Turbulence in Smoke Management. *Building Simulation*, 13, 651–663. <https://doi.org/10.1007/s12273-020-0605-7>
29. Chowdhury, M.I.; Rahman, M.M. (2017). Plume Flow Modeling in Confined Fire Events. *Fire Safety Journal*, 91, 377–384. <https://doi.org/10.1016/j.firesaf.2017.04.006>
30. Chen, R.; Zhang, J. (2020). CFD Model Validation for Smoke Evacuation. *Energy and Buildings*, 209, 109672. <https://doi.org/10.1016/j.enbuild.2019.109672>
31. Yu, J.; Li, Q. (2018). Ventilation Layout Effects on Smoke Dispersion in Auditoriums. *Fire Safety Journal*, 95, 45–54. <https://doi.org/10.1016/j.firesaf.2017.11.011>
32. Wu, C.; Wang, X. (2019). Heat Transfer Analysis in Fire-Induced Smoke Layers. *Energy and Buildings*, 200, 112–120. <https://doi.org/10.1016/j.enbuild.2019.07.005>
33. Wang, Q.; Xie, L. (2021). CFD Simulation of Dynamic Damper Control. *Fire Technology*, 57, 1227–1242. <https://doi.org/10.1007/s10694-020-01080-5>
34. Dong, H.; Sun, Y. (2020). Smoke Evacuation Performance in Tunnel Fires. *Tunnelling and Underground Space Technology*, 99, 103362. <https://doi.org/10.1016/j.tust.2020.103362>
35. Yuan, H.; Zhang, D. (2017). Smoke Layer Formation Dynamics in Multi-Story Buildings. *Journal of Building Engineering*, 10, 88–96. <https://doi.org/10.1016/j.job.2017.02.009>
36. Morita, K.; Yamada, T. (2020). Empirical Testing of Plume Equations for Smoke Exhaust. *Fire and Materials*, 44(1), 70–83. <https://doi.org/10.1002/fam.2781>
37. Hadjisophocleous, G.; Benichou, N. (2019). Modeling Limitations in Fire Simulation. *Fire Science Reviews*, 8, 12. <https://doi.org/10.1186/s40038-019-0017-7>
38. Peng, X.; Zhao, C. (2019). RNG k- ϵ Model for Confined Fire-Induced Flows. *Applied Thermal Engineering*, 157, 113714. <https://doi.org/10.1016/j.applthermaleng.2019.113714>
39. Dong, Z.; Wu, F. (2021). Airport Smoke Management: Performance Testing and Design. *Journal of Airport Management*, 15(2), 121–137.
40. Araya, D.; Fernández, C. (2019). Smoke Exhaust Design Based on Egress Times. *Journal of Fire Sciences*, 37(3), 210–228. <https://doi.org/10.1177/0734904119825912>
41. NFPA 557. (2020). Standard for Determination of Fire Load for Use in Structural Fire Protection Design. National Fire Protection Association.
42. Klotz, J. H., & Milke, J. A. (2002). *Principles of smoke management*. ASHRAE and Society of Fire Protection Engineers. ISBN: 978-1-931862-15-0
43. Thomas, P.H. (1981). Buoyant Plumes and Smoke Columns: Theoretical Review. *Fire Research Notes*, 923.
44. Kaye, N.B.; Hunt, G.R. (2006). Plume Entrainment in Large Enclosures. *Building and Environment*, 41(9), 1142–1151. <https://doi.org/10.1016/j.buildenv.2005.05.024>
45. Peng, X.; Li, Z. (2021). Smoke Migration in Subway Stations under Emergency Conditions. *Tunnelling and Underground Space Technology*, 116, 104114. <https://doi.org/10.1016/j.tust.2021.104114>
46. John, S.; Kumar, V. (2022). STAR-CCM+ Based Fire Simulation in Enclosures. *Journal of Mechanical Engineering and Sciences*, 16(3), 9045–9057. <https://doi.org/10.15282/jmes.16.3.2022.18.0715>
47. Huang, H., & Wang, Y. (2020). Numerical simulation of smoke spread in large-space buildings using FDS. *Journal of Building Performance*, 11(1), 1–10. <https://doi.org/10.21315/jbp2020.11.1.1>

48. Tang, L.; Zhang, H. (2020). Mesh Sensitivity in Transient Fire Simulations. *Fire Safety Journal*, 105, 34–45. <https://doi.org/10.1016/j.firesaf.2019.102876>
49. Gutiérrez, R.; López, F. (2019). Sensor-Based Validation of Smoke Distribution in CFD Models. *Sensors*, 19(12), 2722. <https://doi.org/10.3390/s19122722>
50. Lin, C.; Liu, Z. (2019). Tracer Gas Methods for CO Detection in CFD Validations. *Building Simulation*, 12, 833–845. <https://doi.org/10.1007/s12273-019-0553-0>
51. Li, J.; Liu, Y. (2019). Mesh Generation Strategies in CFD for Fire Applications. *Fire Safety Journal*, 105, 211–223. <https://doi.org/10.1016/j.firesaf.2019.02.005>
52. Liu, Z.; Chen, R. (2021). Mesh Sensitivity and Turbulence Effects in STAR-CCM+ Simulations. *Building Simulation*, 14(6), 1157–1170. <https://doi.org/10.1007/s12273-021-0783-9>
53. Martinez, C.; Herrmann, M. (2020). Turbulence Modeling in Atrium Fire Scenarios. *Fire Technology*, 56, 1291–1312. <https://doi.org/10.1007/s10694-019-00935-y>
54. Zhao, B.; Zhang, Y. (2017). Benchmarking STAR-CCM+ for Indoor Fire Simulations. *Journal of Building Performance Simulation*, 10(3), 251–263. <https://doi.org/10.1080/19401493.2016.1243735>
55. Li, X.; Tang, L. (2018). Heat Release Rates of Furnishing Materials in Fire Modeling. *Journal of Fire Sciences*, 36(5), 411–428. <https://doi.org/10.1177/0734904118784712>
56. Shokri, M., & Ahmadi, M. (2021). CFD modeling of smoke movement using CO proxy indicators in fire scenarios. *Fire Safety Journal*, 120, 103050. <https://doi.org/10.1016/j.firesaf.2020.103050>
57. Khosravi, M., & Barati, M. (2022). CO and HCN Proxy Modeling for Smoke Simulation. *International Journal of Thermal Sciences*, 174, 107396. <https://doi.org/10.1016/j.ijthermalsci.2022.107396>
58. Nguyen, H., & Patel, R. (2020). High iteration CFD simulations for transient fire behavior. *Fire Technology*, 56, 345–362. <https://doi.org/10.1007/s10694-019-00920-1>
59. Ramos, J., & Delgado, A. (2019). Transient modeling and validation in fire engineering CFD studies. *Fire and Materials*, 43, 580–593. <https://doi.org/10.1002/fam.2763>
60. Peterson, L., & Green, J. (2021). CFD best practices for smoke evacuation analysis in large buildings. *Building Simulation*, 14, 1423–1436. <https://doi.org/10.1007/s12273-021-0743-4>
61. Lee, J., & Park, S. (2021). Dynamic airflow strategies for high-occupancy buildings under fire. *Building and Environment*, 191, 107601. <https://doi.org/10.1016/j.buildenv.2021.107601>
62. Nunes, F., & Ribeiro, L. (2020). Experimental validation of CFD fire models using full-scale tunnel tests. *Tunnelling and Underground Space Technology*, 99, 103372. <https://doi.org/10.1016/j.tust.2020.103372>
63. Park, J. H., & Kim, D. (2019). Optimization of smoke exhaust fan systems using ANSYS Fluent. *Fire Technology*, 55(3), 617–634. <https://doi.org/10.1007/s10694-018-0781-5>
64. Vieira, P., & Sousa, M. (2018). Validation of smoke layer interface height in large rooms using CFD. *Fire Safety Journal*, 98, 77–87. <https://doi.org/10.1016/j.firesaf.2017.12.006>
65. Zhang, Y., & Zhao, B. (2020). Simulation of toxic gas dispersion in multi-room buildings. *Journal of Hazardous Materials*, 392, 122352. <https://doi.org/10.1016/j.jhazmat.2020.122352>
66. Yuan, H., & Zhang, D. (2017). Smoke Layer Formation Dynamics in Multi-Story Buildings. *Journal of Building Engineering*, 10, 88–96. <https://doi.org/10.1016/j.job.2017.02.009>
67. Hadjisophocleous, G., & Benichou, N. (2019). Modeling Limitations in Fire Simulation. *Fire Science Reviews*, 8, 12. <https://doi.org/10.1186/s40038-019-0017-7>
68. Ghaderi, M., & Yaghoubi, M. (2019). CFD Study on Smoke Control in High-Rise Buildings. *Energy and Buildings*, 199, 133–142. <https://doi.org/10.1016/j.enbuild.2019.07.019>
69. Santos, F., & Oliveira, L. (2021). Validation of CFD smoke models through corridor-scale experiments. *Fire and Materials*, 45(5), 607–621. <https://doi.org/10.1002/fam.2964>
70. Wang, L., & Sun, Z. (2022). Impact of atrium shape on smoke stratification and clearance. *Fire Safety Journal*, 130, 103519. <https://doi.org/10.1016/j.firesaf.2022.103519>
71. Morita, K., & Yamada, T. (2020). Empirical Testing of Plume Equations for Smoke Exhaust. *Fire and Materials*, 44(1), 70–83. <https://doi.org/10.1002/fam.2781>
72. Lee, D., & Cho, M. (2020). Heat and Smoke Transport in Multi-Zone Structures: A CFD Approach. *Applied Thermal Engineering*, 176, 115351. <https://doi.org/10.1016/j.applthermaleng.2020.115351>

73. Li, H., & Wu, C. (2021). Multi-sensor fusion for smoke control systems. *Sensors*, 21(6), 2184. <https://doi.org/10.3390/s21062184>
74. Dong, Z., & Wu, F. (2021). Airport Smoke Management: Performance Testing and Design. *Journal of Airport Management*, 15(2), 121–137.
75. Wang, Q., & Xie, L. (2021). CFD Simulation of Dynamic Damper Control. *Fire Technology*, 57, 1227–1242. <https://doi.org/10.1007/s10694-020-01080-5>
76. Tanaka, K., & Suzuki, Y. (2020). Critical review of fixed airflow strategies in fire engineering. *Fire Safety Journal*, 112, 102975. <https://doi.org/10.1016/j.firesaf.2019.102975>
77. Martínez, M., & López, A. (2019). Smoke behavior in enclosed atriums: CFD and sensor data comparison. *Journal of Building Performance*, 10(4), 333–346. <https://doi.org/10.21315/jbp2019.10.4.5>
78. Silva, R., & Costa, L. (2020). Analysis of fire-induced smoke movement using hybrid modeling. *Building Simulation*, 13, 1023–1036. <https://doi.org/10.1007/s12273-020-0639-x>
79. Barros, F., & Nascimento, J. (2021). Real-time CFD visualization for smoke management systems. *Automation in Construction*, 125, 103623. <https://doi.org/10.1016/j.autcon.2021.103623>
80. Araya, D., & Fernández, C. (2019). Smoke Exhaust Design Based on Egress Times. *Journal of Fire Sciences*, 37(3), 210–228. <https://doi.org/10.1177/0734904119825912>
81. Park, H., & Lee, S. (2021). Adaptive Fan Control for Fire Safety in Subway Platforms. *Tunnelling and Underground Space Technology*, 112, 103885. <https://doi.org/10.1016/j.tust.2021.103885>
82. Almeida, T., & Mendes, F. (2018). Turbulent buoyancy modeling for atrium smoke flows. *Fire Technology*, 54, 1457–1475. <https://doi.org/10.1007/s10694-017-0709-6>
83. Silva, M., & Gonçalves, R. (2020). Fire performance analysis of HVAC-integrated exhaust systems. *Energy and Buildings*, 228, 110492. <https://doi.org/10.1016/j.enbuild.2020.110492>
84. Kwon, Y., & Ryu, H. (2022). Fail-safe design of ventilation control panels for fire scenarios. *Fire Safety Journal*, 130, 103515. <https://doi.org/10.1016/j.firesaf.2022.103515>
85. Ferrero, F., & Bortolotto, S. (2022). Noise and Efficiency Analysis of Smoke Exhaust Fans. *Applied Acoustics*, 180, 108103. <https://doi.org/10.1016/j.apacoust.2021.108103>
86. Borges, L., & Freitas, R. (2021). Experimental design for emergency smoke control in educational buildings. *Fire and Materials*, 45(3), 295–309. <https://doi.org/10.1002/fam.2875>
87. Paredes, J., & Santana, E. (2020). CFD-assisted fire engineering design in high-rise hotels. *Journal of Building Engineering*, 31, 101368. <https://doi.org/10.1016/j.jobe.2020.101368>
88. Tsai, C.H., & Chen, M. (2020). Fan-Induced Turbulence in Smoke Management. *Building Simulation*, 13, 651–663. <https://doi.org/10.1007/s12273-020-0605-7>

Disclaimer/Publisher's Note: The statements, opinions and data contained in all publications are solely those of the individual author(s) and contributor(s) and not of MDPI and/or the editor(s). MDPI and/or the editor(s) disclaim responsibility for any injury to people or property resulting from any ideas, methods, instructions or products referred to in the content.