fire

MDPI

Review

Smoke and Hot Gas Removal in Underground Parking Through Computational Fluid Dynamics: A State of the Art and Future Challenges

Claudiu Stan [1,*], Ilinca Năstase [1], Florin Bode [2,*] and Răzvan Calotă [1]

[1] CAMBI Research Centre, Technical University of Civil Engineering Bucharest, 021414 Bucharest, Romania; ilinca.nastase@utcb.ro (I.N.); razvan.calota@utcb.ro (R.C.)
[2] AtFlow Research Centre, Technical University of Cluj-Napoca, 400114 Cluj-Napoca, Romania
* Correspondence: claudiu-gabriel.stan@phd.utcb.ro (C.S.); florin.bode@termo.utcluj.ro (F.B.)

Abstract: The proper design and installation of systems that enable the efficient control and removal of smoke and hot gases in underground parking facilities are necessary for protecting the public and property in the event of a fire. This paper discusses how studies using Computational Fluid Dynamics (CFD) related to smoke venting have contributed to improving fire safety in underground parking facilities. As vehicle fire incidents continue to rise globally, particularly in regions with a high density of underground parking, the need for comprehensive measures to mitigate these incidents has become increasingly urgent. This paper examines the applicability of CFD as a tool to address the challenges of smoke control in underground car parks, including those caused by fires involving electric vehicles. CFD application under various fire scenarios and ventilation strategies allows for identifying more effective smoke removal solutions, improving the protection of occupants and property. However, despite the potential of CFD simulations to enhance fire safety and smoke exhaust efficiency in underground parking, it is important to recognize the limitations of these simulations, particularly in dealing with the complex challenges posed by electric vehicle fires.

Keywords: fire safety; ventilation strategies; CFD simulation; smoke extraction; underground parking

check for updates

Citation: Stan, C.; Năstase, I.; Bode, F.; Calotă, R. Smoke and Hot Gas Removal in Underground Parking Through Computational Fluid Dynamics: A State of the Art and Future Challenges. *Fire* **2024**, *7*, 375. https://doi.org/10.3390/fire7110375

Academic Editor: Zihe Gao

Received: 25 September 2024
Revised: 17 October 2024
Accepted: 22 October 2024
Published: 24 October 2024

Copyright: © 2024 by the authors. Licensee MDPI, Basel, Switzerland. This article is an open access article distributed under the terms and conditions of the Creative Commons Attribution (CC BY) license (https://creativecommons.org/licenses/by/4.0/).

1. Introduction

Underground facilities pose unique challenges for fire safety due to their distinct technical features, which influence fire spread and smoke behavior differently than in above-ground buildings. In recent years, CFD has proved to be an indispensable tool in the design and evaluation of effective smoke extraction systems, offering detailed insights into the dynamics of smoke movement and enabling the optimization of ventilation strategies [1].

Incidents of fires in underground parking facilities are frequently attributed to various problems associated with the vehicles parked within these confined spaces. Factors such as electrical malfunctions, overheating engines, overheating batteries, or fuel leaks in parked vehicles can significantly increase the risk of igniting a fire [2]. The enclosed nature of underground parking spaces worsens the situation by facilitating the rapid accumulation of combustible gases and limiting the dispersion of smoke and heat [3]. The characteristics of the smoke depend significantly on the types of vehicles involved. For instance, traditional internal combustion engine vehicles typically produce smoke with a high concentration of carbon monoxide, hydrocarbons, and soot when they catch fire. These emissions include hydrochloric acid (HCl), sulfur dioxide (SO_2), volatile organic compounds (VOCs) such as benzene, polycyclic aromatic hydrocarbons (PAHs), and polychlorinated dibenzo-p-dioxins/polychlorinated dibenzofurans (PCDDs/PCDFs) [4]. In contrast, electric vehicles, which are becoming increasingly common, generate smoke that may contain toxic gases like hydrogen fluoride due to the combustion of lithium-ion batteries [5]. Furthermore, the materials used in the construction of different vehicle models, such as plastics, metals, and

synthetic fibers, also influence the composition and behavior of the smoke. These variations necessitate tailored smoke management strategies to effectively address the diverse smoke characteristics and ensure optimal safety in underground parking environments.

In recent years, underground parking fires have caused considerable damage and disruption on several occasions worldwide. In Paris, France, in October 2014 [6], a fire began on the fourth level of the seven-story Edouard VII car park behind the famous L'Olympia Theater. On this occasion, 50 cars were destroyed, and the busy road was closed for hours while part of the center of Paris was affected by thick, choking smoke. In the heart of many European cities, underground car parks are a common solution to the challenge of urban space constraints. These subterranean complexes serve as vital infrastructure, accommodating the vehicular demands of densely populated areas. However, this convenience has been shadowed by an alarming trend in recent years: a series of major fires within these parking facilities. The incidents have not only highlighted vulnerabilities in safety measures but also posed significant risks to emergency response teams. The nature of fires in these underground settings is particularly perilous due to the extreme temperatures that can rapidly develop, fueled by confined spaces and abundant combustible materials found in vehicles, such as petrol, oil, and synthetic materials in car interiors. Such conditions often escalate beyond the threshold where firefighting operations can be safely conducted. Firefighters, facing the dual challenges of intense heat and limited accessibility, frequently find it impossible to approach the core of the blaze to properly apply extinguishing agents. The risk of structural collapse, toxic smoke inhalation, and the potential for sudden fire growth further complicate their efforts, making it exceedingly hazardous to attempt suppression operations [6]. Another fire at Luton Airport's [7] parking lot destroyed more than 1500 vehicles, with early indications suggesting it was an accidental incident resulting from a malfunction in a diesel-powered vehicle. The report stated that most of the affected vehicles cannot be restored and has provided registration details to the Motor Insurers' Bureau [7].

Several cars went up in flames one early morning in 2023 after a fire broke out in a covered garage at the Armon Hotel on Summer Street in Stamford. Eight vehicles were destroyed, and another eight were significantly damaged; the intense heat raised concerns about the structural integrity of the parking facility. Also, the structure of the garage was compromised in the blaze [8,9]. These incidents show the increasing vulnerability of underground parking facilities to catastrophic fires, driven by a combination of structural design limitations and the inherent risks associated with the materials found in modern vehicles.

Underground parking facilities, by their enclosed nature, present significant challenges in controlling fire size and spread. The lack of natural ventilation and complex layouts allows fires to grow rapidly, making containment difficult. Consequently, more attention is needed on assessing fire size and spread dynamics, as this influences both smoke propagation and the effectiveness of firefighting efforts.

It is well known that to minimize environmental pollution, the proportion of electric cars is increasing compared to other types of cars [10].

Almost 14 million new electric cars were registered globally in 2023, bringing their total number on the roads to 40 million. Electric car sales in 2023 were 3.5 million higher than in 2022, indicating a 35% year-on-year increase (Figure 1). This is more than six times higher than in 2018 just 5 years earlier. In 2023, there were over 250,000 new registrations per week, which is more than the annual total in 2013 ten years earlier. Electric cars accounted for around 18% of all cars sold in 2023, up from 14% in 2022 and only 2% five years earlier in 2018. These trends indicate that growth remains robust as electric car markets mature. Battery electric cars accounted for 70% of the electric car stock in 2023 [10].

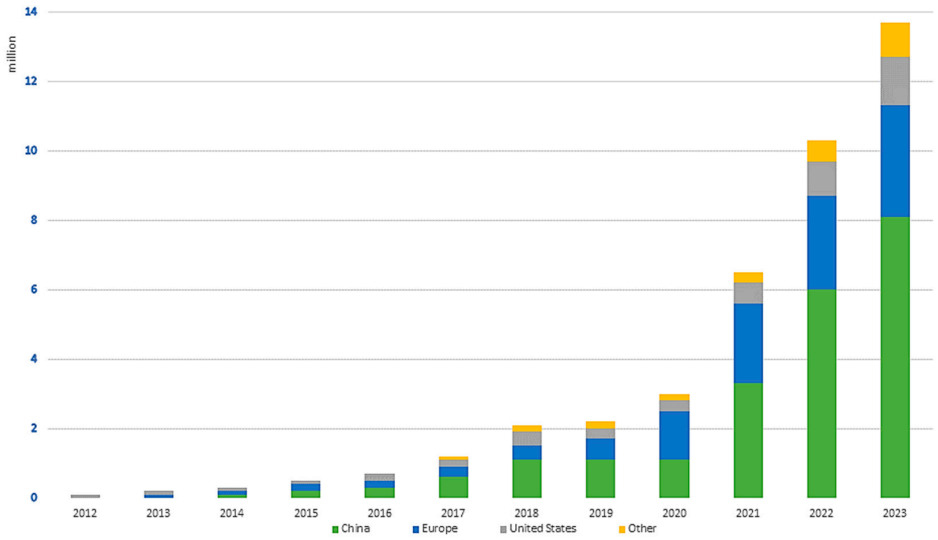


Figure 1. Evolution over time of electric car sales 2012–2023 [10].

Electric car sales remained strong in the first quarter of 2024, surpassing those of the same period in 2023 by around 25% to reach more than 3 million (Figure 2). This growth rate was like the increase observed for the same period in 2023 compared to 2022. Most of the additional sales came from China, which sold about half a million more electric cars than over the same period in 2023. In relative terms, the most substantial growth was observed outside of the major EV markets, where sales increased by over 50%, suggesting that the transition to electromobility is picking up in an increasing number of countries worldwide [10].

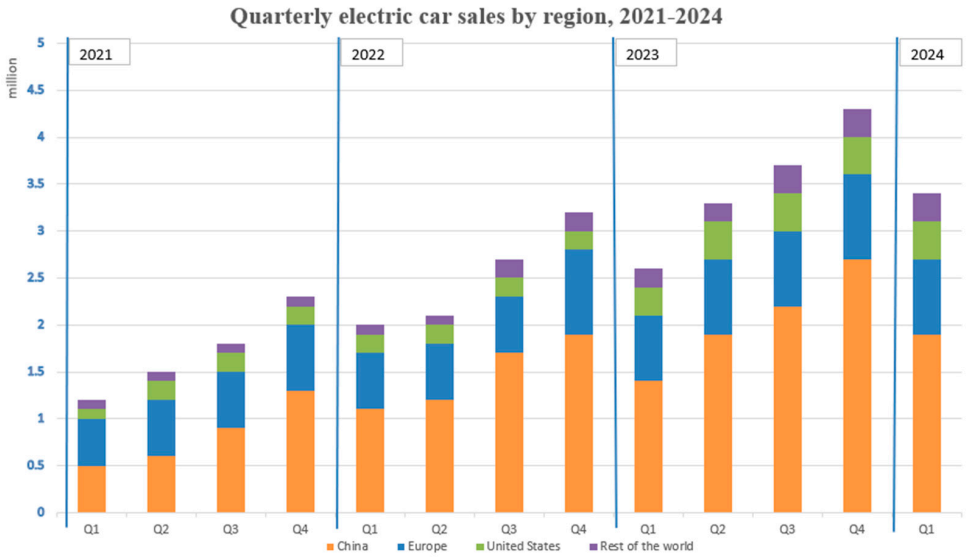


Figure 2. Quarterly electric car sales by region 2021–2024 [10].

Although fires involving battery electric vehicles (BEVs) can present a higher risk to life safety and property protection compared to those caused by vehicles with internal combustion engines (ICEVs) within parking facilities, the research on emission factors and fire characteristics of BEVs is not as extensive as that of conventional ICEVs [11]. Due to these changes, the impact of BEVs on ventilation design for underground parking garages must be considered carefully. The results of the current literature studies show that although exhaust emissions have been eliminated, BEVs still generate non-exhaust particles, which account for more than 85% of the total traffic particles [11]. Due to the large vehicle weight, non-exhaust particles produced by BEVs may be comparable to the total

traffic particles emitted by their ICEV counterparts. The average heat release rate of BEV fire is similar to that of their ICEV counterparts, but the hydrogen fluoride concentration generated is higher, and water is still suitable for eliminating BEV fire [12]. At present, the impact of BEVs on underground parking garage ventilation design is not yet evident [13].

When a fire is started in an EV [14] in an underground facility, the damage can destroy the basement and require complete renovation of the building.

Firefighters have expressed concerns that once an electric vehicle catches fire (Figure 3), extinguishing it becomes near impossible, with the blaze burning at such elevated temperatures that it threatens the entire structure, regardless of its height. In response to these risks, Belgium authorities are reconsidering the admission of electric cars. They have announced intentions to enact local legislation that would prohibit electric vehicles from parking in underground facilities. Instead, owners of electric vehicles will be required to park on the street, even if they possess paid parking spaces below ground [14].



Figure 3. Belgium, 2022, after an EV caught fire in a parking lot [14].

In a recent development that underscores the unique challenges posed by electric and hybrid vehicles, a decision has been made to prohibit these vehicles from parking in underground garages. Following a major incident where a vehicle fire caused a five-month closure of an underground parking facility, Michael Kuhnlein, from the civil engineering department, highlighted the necessity of new regulations. These are needed due to the challenges of fires posed by electric and hybrid vehicles, which are difficult to extinguish because of their lithium batteries. The structural limitations of underground garages, such as insufficient height, further complicate firefighting efforts. This policy adjustment underscores the need for continuous updates to infrastructure and safety protocols to keep pace with advances in automotive technology [15]. Stricter regulations on BEV parking locations, especially in enclosed spaces like underground garages, are essential due to concerns related to fire safety. Research on the fire characteristics of BEVs, as well as the development of more effective smoke exhaust methods, can significantly enhance the safety measures in place. Understanding the emissions impact of BEVs, including both exhaust and non-exhaust emissions, is vital for comprehensive regulations [16].

A Fire Dynamics Simulator is a commonly used tool in fire engineering to numerically simulate fires. This software uses an approximate form of Navier–Stokes equations, appropriate for low Mach numbers [17]. As a result of the widespread adoption and implementation of Computational Fluid Dynamics (CFD) software, numerous research efforts have been directed towards developing numerical methods capable of modeling various scenarios like the combustion mechanisms of a vehicle on fire in a tunnel, the characteristics of fire-induced smoke, including its properties, shape, movement, and interactions with the ceiling and other structures, the key factors that influence the ventilation system, such as heat release rate, reference length, and other relevant parameters.

Computational Fluid Dynamics (CFD) is a valuable tool in enhancing fire safety, particularly in improving smoke exhaust efficiency in underground parking environments. CFD numerical simulations are increasingly used to assist in the design of smoke management

systems in underground structures to ensure compliance with fire safety regulations [18]. By utilizing CFD numerical modeling, fire safety engineers can analyze and predict fire behavior, smoke propagation, and the effectiveness of ventilation systems in underground car parks [13].

A notable case study that illustrates the effective application of the CFD method in managing EV fire incidents is the study conducted by Dorsz and Lewandowski [19], which analyzed fire hazards associated with electric vehicles in enclosed spaces. Their study utilized CFD numerical methods to model fire scenarios, comparing the fire dynamics of electric vehicles with internal combustion engine vehicles (ICEVs) in confined spaces such as underground garages, and provided a comprehensive performance-based analysis of safety in car parks under electric vehicle fire conditions. Their case study employed fire simulations to predict conditions for safe evacuation and the operational parameters for fire brigade interventions during an EV fire. CFD is used to simulate the environmental conditions during a fire, accounting for the heat release rate (HRR), smoke generation, and the propagation of fire in confined spaces. By providing accurate predictions of how a fire develops, CFD helped in understanding the rate at which the temperature increased and smoke spread. These data allow fire safety systems such as ventilation and sprinklers to be optimized to control the situation effectively and provide safe evacuation paths. Using CFD, different scenarios were modeled to design or optimize systems like ventilation fans, exhaust systems, and sprinkler systems to mitigate fire hazards. For instance, the document describes a case where a small underground garage was equipped with a fire ventilation system. CFD simulations predicted the effectiveness of this system in maintaining visibility and safe temperatures for both evacuees and firefighting teams during an EV fire. The simulations demonstrated how CFD could be utilized to model smoke movement and heat distribution, thereby informing the design of fire safety protocols and emergency response plans. This integration of CFD into fire safety planning is crucial, as it allows for a proactive approach to managing the risks associated with EV fires. The results highlighted significant differences in fire behavior, smoke production, and heat release rates, which are critical for developing effective evacuation strategies and fire suppression techniques in such environments [19].

Research has demonstrated that CFD simulations are particularly important in evaluating the performance of smoke prevention and exhaust systems in various buildings, including underground garages, metro stations, and tunnels [20]. Additionally, CFD simulations have been employed to compare different designs of fire ventilation systems, optimizing the layout and efficiency of smoke extraction mechanisms in underground parking facilities [21]. The effectiveness of smoke control systems in underground structures is a critical aspect of fire safety design, with factors such as heat release rate and ventilation system design influencing the efficiency of smoke extraction [22]. Furthermore, CFD simulations have been utilized to assess the ventilation efficiency of mechanical ventilation systems in underground parking lots, emphasizing the significance of proper ventilation in ensuring smoke control during fire incidents [23].

CFD studies allow for the detailed analysis of airflow, heat transfer, and smoke propagation within confined spaces, enabling engineers and safety experts to predict how fires will evolve in various scenarios. By simulating different fire conditions and ventilation strategies, CFD helps to optimize the design and installation of smoke control systems, which can improve the evacuation of toxic gases and the dissipation of heat.

Furthermore, CFD can assist in identifying areas within a parking structure that are most vulnerable to fire spread or smoke accumulation, allowing for the precise placement of exhaust vents and fire suppression systems. This predictive capability is particularly important in underground environments, where limited access and confined spaces can hinder traditional firefighting efforts. By preemptively addressing these challenges through CFD simulations, it becomes possible to implement more effective fire mitigation strategies, lowering both the likelihood and severity of such incidents. Additionally, CFD studies

contribute to enhancing the safety of emergency response teams by providing insights into safer and more effective routes for firefighting operations in these high-risk environments.

To provide a clearer understanding of the research landscape in the field of fire safety and smoke management in underground parking facilities, it is important to first explore the key themes and trends that have shaped this domain in the last year. By analyzing the existing literature and examining the frequency of certain terms and topics in scientific publications, we can gain valuable insights into how this field has evolved over time.

For example, Figure 4 presents a word cloud displaying key terms extracted from the literature cited in this research. This visual depiction highlights the diverse facets of the research domain, encompassing a spectrum of topics beyond smoke exhaust and management, extending to practical applications.



Figure 4. Word cloud of the keywords extracted from the bibliographical references cited in the current study.

While the word cloud incorporates seminal works and standards that trace the historical evolution of smoke exhaust from car parking using Jet fans and related CFD studies, the primary focus of this paper lies in contemporary research that offers fresh perspectives and avenues relevant to our specialized field of study but the increasing frequency of use of CFD by researchers in their publications in the last 25 years is also highlighted by the graph from Figure 5, with the most notable increase occurring in the last five years.

This rise underscores growing concerns and the intensified focus on improving fire safety measures, particularly in response to the evolving challenges posed by modern technologies and vehicle types, and this is proof that CFD has become increasingly prevalent in recent years. The red column from Figure 5 represents the number of scientific articles published on the same topics as the first column, but with the addition of electric vehicles, this shows that in recent years, electric vehicles became a significant research topic, but it still constitutes a smaller subset of the overall research in CFD, fire, numerical simulation, and smoke. However, the increasing trend in the red column suggests that the intersection of these topics with EVs is becoming more significant due to the growing importance of EVs in modern technology and the associated fire safety concerns.

The article aims to explore the application of Computational Fluid Dynamics (CFD) simulations in addressing challenges related to smoke evacuation in underground car parks during fire incidents, including those involving electric vehicles. It discusses the capabilities and limitations of using CFD simulations for modeling smoke movement, heat distribution, and evacuation dynamics in such environments [24]. The study highlights how CFD simulations can be used to analyze smoke evacuation strategies, optimize smoke management systems, and enhance safety measures in underground car parks. It emphasizes the importance of accurate input parameters and computational resources for detailed simulations [24]. Additionally, the article discusses the strengths of CFD simulations in replicating environmental conditions during car fires to assess evacuation procedures, firefighting operations, and structural safety in underground car parks [25].

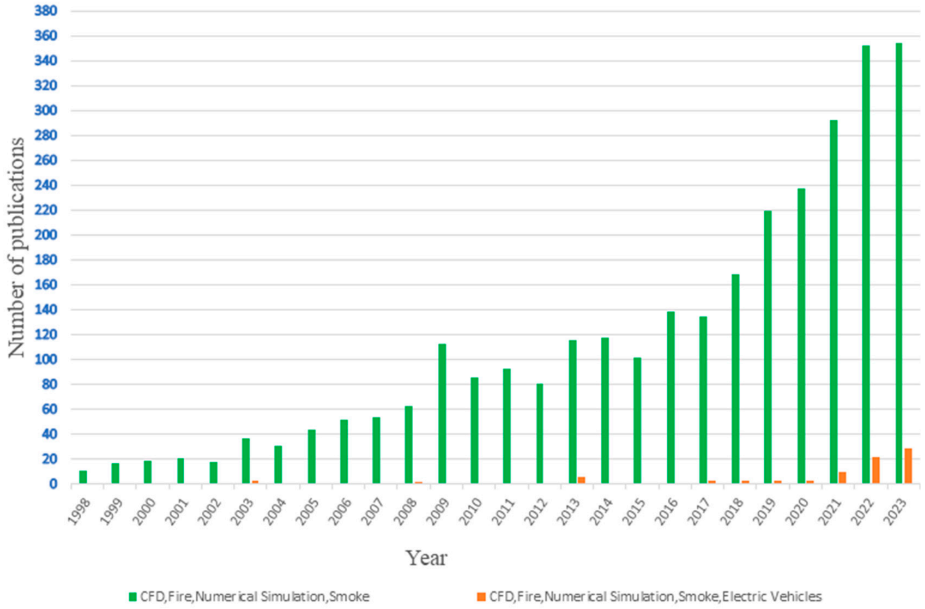


Figure 5. Scientific articles published in Scopus annually in the last 25 years containing the words:CFD, fire, numerical, simulation, smoke (with/without electric vehicles).

Figure 6 displays a comparison between studies containing the words CFD, fire, and smoke in various locations like tunneling and car parks. While both areas are important, the higher number of publications in the "CFD, Fire, Smoke, tunneling" category suggests a greater focus on understanding and mitigating risks in tunneling environments, reflecting their complexity, regulatory demands, and potential impact on safety and infrastructure. Conversely, the lower number of studies on car parks indicates a need for increased research to address specific challenges in this area, such as ventilation and fire safety in confined and multi-story car parks.

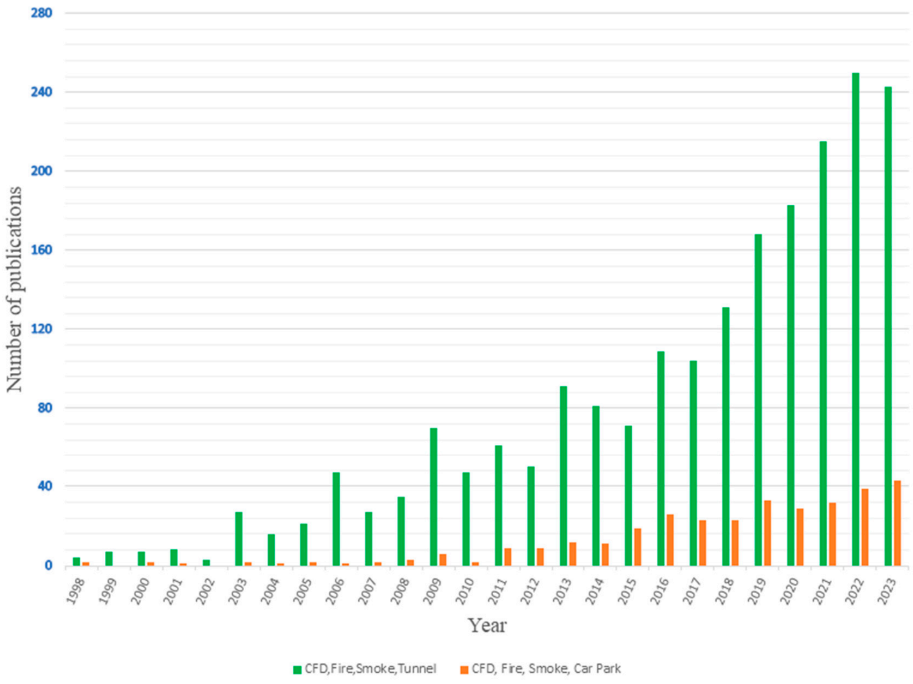


Figure 6. The number of scientific articles published in the Scopus Database annually in the last 25 years containing the words scientific; CFD, fire, smoke, and (tunnel or car park).

The workflow of this article is presented below in Figure 7.

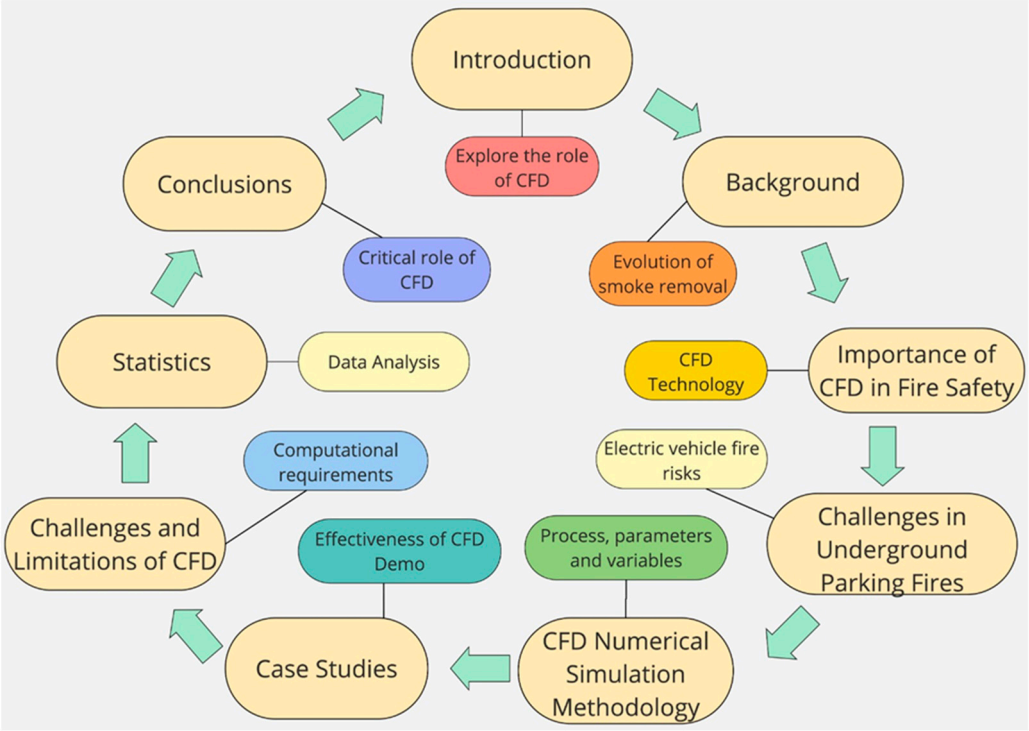


Figure 7. The workflow of this article.

The scope of this paper is to comprehensively review the application of Computational Fluid Dynamics (CFD) simulations in addressing the complex challenges of smoke evacuation in underground car parks, with a specific focus on the limitations and capabilities of this method. The paper aims to critically evaluate the excessive dependence on CFD simulations, considering the accuracy of data input, applied physical models, user experience, and model performance. Additionally, it seeks to provide insights into the challenges posed by electric vehicle fires and their impact on smoke evacuation strategies in underground parking structures.

The paper aims to examine the limitations of CFD simulations in predicting fire dynamics and smoke removal strategies in underground car parks. It also seeks to assess the capabilities and drawbacks of CFD simulations in addressing smoke evacuation challenges, including those from electric vehicle fires, and offer recommendations for improving their reliability. Additionally, the paper evaluates the effectiveness of integrating impulse fans with ventilation systems for enhancing air quality and safety during fire evacuations in underground parking areas. It discusses the significance of empirical validation in verifying the accuracy of CFD simulations, particularly in predicting fire dynamics related to electric vehicle fires. Finally, it provides insights into the limitations and capabilities of CFD simulations in smoke evacuation in underground car parks, emphasizing their practical applicability in real-world scenarios.

The article provides an overview of leveraging CFD simulations to address smoke evacuation challenges in underground car parks, particularly focusing on electric vehicle fires. By examining both the capabilities and limitations of CFD simulations, the article contributes to enhancing fire safety measures in underground structures.

The accuracy of CFD simulations is limited by the precision of the data input, the applied physical models, and the performance of the model itself. This is particularly relevant when simulating electric vehicle fires, as the unknown risk profile and fire characteristics associated with lithium-ion batteries pose significant challenges to accurate simulation and prediction of fire events.

2. Background

The unique challenges presented by underground facilities require specialized fire safety strategies that go beyond those typically employed in conventional buildings. This includes the development of more sophisticated smoke management systems, the implementation of advanced evacuation protocols, and the use of innovative firefighting techniques tailored to the complexities of underground environments. Moreover, the design and layout of new underground facilities must take into consideration these fire safety challenges, incorporating features that enhance ventilation, improve egress options, and facilitate effective emergency response.

There are two main types of smoke exhaust systems for underground parking garages: conventional ventilation systems and ductless systems with jet fans. In recent years, ductless systems with jet fans have become increasingly popular due to their ability to effectively manage smoke spread.

Conventional ventilation systems typically use ducts and air diffusers to distribute fresh air throughout the garage and remove smoke from designated exhaust areas. These systems have the advantages of known performance standards, are widely used in many facilities, and can remove smoke directly through designated exhaust ducts, ensuring targeted smoke removal. However, these systems can be less effective in complex environments like underground parking garages, where multiple lanes, driveways, and utility systems can interfere with airflow. The complexity of the garage layout can reduce the uniformity of air distribution and make it difficult to remove pollutants effectively. Another disadvantage of conventional ventilation systems is that they are difficult to implement in parking areas with complex geometries, resulting in uneven smoke removal and higher installation and maintenance costs due to extensive piping requirements.

Ductless systems with jet fans offer a more effective solution for smoke exhaust in underground parking garages. These systems use high-velocity jet fans to direct a concentrated stream of air towards the smoke source, effectively dispersing and removing the smoke from the garage. The localized nature of jet fans allows for more precise control of smoke spread, ensuring that clean air is distributed to the rest of the garage.

Due to their superior performance in complex environments, ductless systems with jet fans are increasingly being adopted by designers and engineers for smoke exhaust in underground parking garages. These systems provide a more effective and efficient way to regulate smoke spread and ensure the safety of occupants. Jet fans, sometimes referred to as tunnel ventilation fans or impulse fans, are specifically engineered fans tailored to various applications, particularly in tunnels, underground parking garages, and enclosed spaces where airflow control and pollution prevention are paramount. These compact fans are typically installed on walls or ceilings and can be mounted in either a vertical or horizontal orientation, contingent on the specific application and airflow requirements. Horizontal mounting is prevalent in parking garages, while vertical mounting is favored in tunnels. The installations of jet fans come with a high energy consumption during operation as large volumes of air must be moved continuously to manage smoke and a higher noise level due to the operation of multiple fans. Another disadvantage is that more advanced control systems are needed to ensure coordinated fan operation and optimal smoke movement, and the performance can be compromised in certain scenarios if not correctly designed for airflow management.

The evolution of smoke removal strategies in underground parking facilities has progressed significantly over the years, culminating in the adoption of Computational Fluid Dynamics (CFD) simulations. Initially, smoke control in underground parking areas relied on traditional mechanical ventilation systems aimed at extracting smoke and heat during fire incidents. The research made by B. Merci and M. Shipp, 2013 [12] presents a detailed study on smoke and heat control in large car parks using Computational Fluid Dynamics (CFD) simulations. The research focuses on the effectiveness of forced mechanical horizontal ventilation in managing smoke during a car park fire. The study aims to provide scientific support for developing standards and guidelines for designing Smoke and Heat

Control (SHC) systems in car parks. The methodology used involves simulations of full-scale car park fire experiments where the fire is modeled as a well-controlled liquid pool fire. The result of the study confirms that CFD can accurately predict smoke patterns, particularly the smoke back-layering distance, under various setups. They conclude that the CFD simulations provide a valuable tool for understanding and designing effective smoke control systems in car parks, but the complexity of smoke dynamics in large car parks underscores the need for detailed modeling and analysis to ensure effective smoke management in fire scenarios [12].

However, as research and technology advanced, there was a shift towards more sophisticated approaches integrating CFD simulations to enhance the effectiveness of smoke removal strategies. CFD simulations of full-scale configurations in large car parks provided valuable insights into the impact of structural elements like beams on ventilation system performance, emphasizing the need for tailored solutions in underground spaces [26].

The utilization of CFD simulations allowed engineers to model and optimize ventilation systems specifically tailored to the challenges posed by underground environments. Studies began to focus on the relationship between ventilation velocity and smoke back layering distance in large, closed car parks, highlighting the importance of understanding airflow dynamics for efficient smoke control [24].

The development of jet fan ventilation systems further revolutionized smoke removal strategies in underground parking areas. These systems were designed not only for carbon monoxide removal under normal conditions but also for smoke extraction during emergency scenarios such as fires [27]. By incorporating CFD numerical simulations, researchers can analyze the effectiveness of jet fan ventilation systems in improving indoor air quality and enhancing smoke control measures in underground parking facilities [27].

The evolution of smoke removal strategies in underground parking facilities has seen a significant advancement with the integration of CFD simulations. By using the CFD technique, engineers can design and optimize ventilation systems to address the unique challenges of underground environments, improving fire safety and ensuring the well-being of occupants and emergency responders in underground structures.

3. Importance of CFD in Fire Safety

The Computational Fluid Dynamics (CFD) technique has become an important tool in fire safety analysis, especially in the context of smoke removal in underground parking spaces. CFD allows engineers to simulate and analyze complex fluid flow and heat transfer phenomena virtually. In fire safety, CFD numerical simulations help predict smoke movement, temperature distribution, and airflow patterns during fire incidents, aiding in designing and optimizing smoke control systems. The use of CFD in underground parking spaces involves modeling smoke behavior during a fire event. By inputting parameters like fire location, heat release rate, ventilation system design, and parking structure geometry, CFD simulations can predict smoke propagation within the interest domain. This information is necessary for designing effective smoke removal strategies to ensure the safety of occupants and emergency responders in underground parking facilities.

Studies carried out by Deckers [26,28] have utilized CFD simulations to explore smoke control in large car parks. These simulations offer insights into the impact of smoke and heat control systems on smoke patterns and ventilation efficiency during fire incidents. Additionally, research by Węgrzyński [29] emphasizes the importance of smart smoke control solutions based on adaptive fan performance, highlighting the effectiveness of CFD-based design approaches in optimizing smoke ventilation systems. The study identified the adaptive performance of smoke exhaust fans based on internal conditions as a favorable approach for spaces with limited smoke reservoir size or duct space. In the domain of fire safety in car parks, ventilating systems for indoor air quality and smoke exhaust were evaluated, highlighting the critical role of ventilation in managing smoke during fire incidents [29].

Moreover, CFD simulations have been very used in evaluating the performance of jet fan ventilation systems in underground parking areas for smoke extraction during emergencies. Through numerical studies researchers can assess different ventilation configurations and flow rates effectiveness in enhancing carbon monoxide removal and smoke control in underground parking environments.

As stated earlier, CFD is a valuable tool for analyzing and optimizing smoke removal strategies in underground parking spaces. By utilizing CFD numerical simulations, engineers can design efficient ventilation systems that reduce the risks associated with smoke accumulation during fire incidents, enhancing fire safety in underground parking facilities.

In the context of CFD numerical simulations for the removal of smoke and hot gases in underground parking facilities, three primary numerical methods can be used: RANS (Reynolds-Averaged Navier–Stokes), LES (Large Eddy Simulation), and DNS (Direct Numerical Simulation).

The RANS method is one of the most used techniques in CFD due to its computational efficiency. It involves averaging the Navier–Stokes equations over time, which allows for the resolution of turbulent flows without needing to simulate every small eddy in the flow. This method simplifies the computational process and reduces the required resources, making it suitable for large-scale simulations such as those needed for smoke and hot gas removal in underground parking structures [30]. The RANS method has several drawbacks when applied to fire simulation and smoke management.

First, RANS provides limited resolution of turbulent flows since it involves time-averaging the Navier–Stokes equations, which can result in a less accurate representation of smoke and hot gas behavior. Second, it relies on turbulence models, introducing potential errors and uncertainties in the simulation. These models may not be suitable for all fire scenarios, which affects result accuracy. Additionally, RANS is sensitive to initial and boundary conditions specified in the simulation. Any inaccuracies in these conditions can significantly impact the results. Moreover, due to its averaging nature, RANS cannot capture high-frequency fluctuations and fine details of turbulent flow, which are especially important for understanding complex fire dynamics.

LES is a more advanced technique that provides a better representation of turbulent flows than RANS. It involves resolving the larger eddies directly while modeling the smaller, sub-grid scale eddies. This method strikes a balance between accuracy and computational cost, offering more detailed insights into the flow dynamics and turbulence structures. LES is particularly useful for simulating complex fire scenarios in underground parking, where the accurate depiction of turbulence and smoke behavior is necessary for effective smoke management strategies [31–33].

The LES method, while more advanced than RANS, also has notable drawbacks for fire simulation and smoke management. Primarily, LES demands significantly higher computational resources because it directly resolves larger turbulent eddies while modeling the smaller ones, resulting in greater computational intensity and longer simulation times. Additionally, LES requires more sophisticated algorithms and expertise to implement accurately, usually making the setup more complex. The method is also extremely sensitive to grid resolution, so insufficient grid detail can lead to inaccurate results, further increasing computational costs. While LES improves accuracy in depicting turbulence, it still relies on sub-grid scale models for smaller eddies, which can introduce uncertainties. Moreover, LES is more sensitive to boundary conditions, necessitating precise specifications to ensure reliable outcomes. Due to these computational and practical constraints, LES might not be feasible for very large-scale problems or scenarios requiring real-time simulation, limiting its applicability despite its detailed insights into turbulent flows.

DNS is the most detailed and computationally intensive method, as it resolves all scales of turbulence by solving the Navier–Stokes equations directly without any modeling approximations. This method provides the highest level of accuracy in simulating fluid flow and turbulence. However, due to its huge computational demands, DNS is limited to smaller, more fundamental studies rather than large-scale practical applications and is

not feasible for solving practical problems in fire or wind engineering. In the context of underground parking facilities, DNS can be used for detailed analysis of specific scenarios to improve the understanding of smoke and gas behavior [33].

Several software packages are widely used for CFD simulations, each offering various capabilities for implementing RANS, LES, and DNS methods. Some of the most used CFD software are presented in Table 1.

Table 1. Most used CFD software.

Software	RANS (Reynolds-Averaged Navier–Stokes)	LES (Large Eddy Simulation)	DNS (Direct Numerical Simulation)	Special Features
ANSYS Fluent [34]	YES	YES	Limited	Robustness, wide range of applications
ANSYS CFX [35]	YES	YES	NO	Strong in complex fluid and multiphase flows
OpenFOAM [36]	YES	YES	Limited	Open-source, highly customizable
STAR-CCM+ [37]	YES	YES	NO	Comprehensive fluid dynamics tools
COMSOL [38]	YES	YES	Limited	Multiphysics integration, versatile
FDS/PyroSim [39]	NO	YES	Limited (with fine grid resolution)	Specialized in fire dynamics simulations

These software tools enable researchers and engineers to perform detailed CFD simulations to enhance fire safety measures and improve smoke and hot gas removal strategies in underground parking facilities.

CFD simulations are invaluable for predicting how fire size affects smoke movement and heat distribution in underground parking garages. By incorporating fire size as a critical factor in simulations, engineers can optimize ventilation systems and improve evacuation strategies. This is particularly important for controlling the spread of fire, which can drastically affect air quality and visibility, impairing both occupant safety and firefighting operations.

4. Challenges in Underground Parking Fires

Challenges in underground parking fires present complex scenarios that require a deep understanding of fire spread, smoke movement, and ventilation systems to ensure the safety of occupants and effective firefighting strategies.

The challenges posed by underground spaces for fire safety are unique due to technical characteristics that significantly influence fire propagation differently than in conventional buildings. Underground environments, such as parking facilities and tunnels, present distinct challenges due to limited ventilation, complex geometries, and restricted egress options. These factors contribute to the rapid spread of smoke and heat during fires, leading to greater risks for occupants and emergency responders [40].

In such environments, fire can escalate rapidly, overwhelming traditional fire suppression systems and putting in danger both occupants and emergency personnel. Without a precise understanding of smoke behavior and ventilation efficiency, these incidents can get out of control in several minutes, making advanced predictive models essential for safety planning.

4.1. Limited Ventilation

The ventilation systems in underground spaces are designed primarily to manage air quality and remove exhaust gases under normal operating conditions. However, in the event of a fire, these limited ventilation capabilities can become a liability. The restricted

airflow may hinder the dispersion of smoke and heat, causing them to accumulate rapidly. This not only makes the environment more dangerous for occupants but also hinders firefighting efforts.

The impact of fire locations on the performance of ventilation systems in underground car parks has been studied, emphasizing the role of ventilation in preventing smoke spread and maintaining visibility for firefighters [41].

4.2. Complex Geometries

The layout and design of underground spaces often involve complex geometry, including long corridors, multiple levels, and confined spaces, which can influence fire behavior in unpredictable ways. Such complexities can lead to uneven heat distribution, creating hotspots that are difficult to manage. Additionally, the intricate layouts can disorient occupants and emergency responders, hampering evacuation and rescue efforts. Complex geometries in underground parking fires present unique challenges that require a comprehensive understanding of fire dynamics, smoke movement, and ventilation systems. The interaction of various flow types, such as large-scale background flow, sink flow, and vortex circulation, becomes intricate as the geometry of the fire evolves [42].

4.3. Restricted Egress Options

Unlike above-ground buildings, which may offer multiple exit routes, underground spaces are typically characterized by few entry and exit points. In a fire, these restricted egress options can lead to bottlenecks, slowing evacuation processes. The limited number of escape routes can also increase the risk of exposure to smoke and heat for occupants trying to flee the area. When a fire occurs in an underground parking facility, especially in areas like transformer rooms adjacent to cable tunnels that serve as main egress lanes, the high fire intensity can make it extremely difficult for evacuees to escape safely [43].

4.4. Rapid Spread of Smoke and Heat

The combination of limited ventilation, complex geometries, and restricted egress options in underground spaces can contribute to the rapid spread of smoke and heat during a fire. Smoke, being less dense than air, tends to rise; however, in an underground setting, it can quickly fill the entire space, lowering visibility and air quality. Heat, too, can build up more quickly in these enclosed environments, increasing the risk of flashover and further complicating firefighting and rescue operations. The interaction of smoke and heat in underground structures can lead to hazardous conditions, impeding evacuation efforts and endangering occupants. Studies have shown that with increasing fire power, smoke accumulation accelerates, leading to elevated heating rates and reduced visibility, making evacuation more challenging [44].

The spread of fire in underground parking facilities can lead to catastrophic consequences due to the confined environment, allowing heat and toxic smoke to accumulate quickly. Given that fire size can escalate rapidly in these conditions, it is essential to consider how fire spread affects ventilation system performance and evacuation plans. Additionally, with the increasing presence of electric vehicles, which can burn at higher temperatures, a deeper focus on fire size and its containment is critical for developing robust safety protocols.

4.5. EV Fires Challenges

Current data and research suggest that while electric vehicle (EV) fires are currently rare, they could become more common as the number of EVs increases and these vehicles age. A study by EV FireSafe reports that between 2010 and June 2023, there were 488 light-duty EV fires worldwide, with 393 of these (78%) being confirmed as fires involving EV lithium-ion batteries. The study also indicates that as the EV market continues to expand, the frequency of EV fires is likely to rise. This is illustrated in Figure 8 [45].

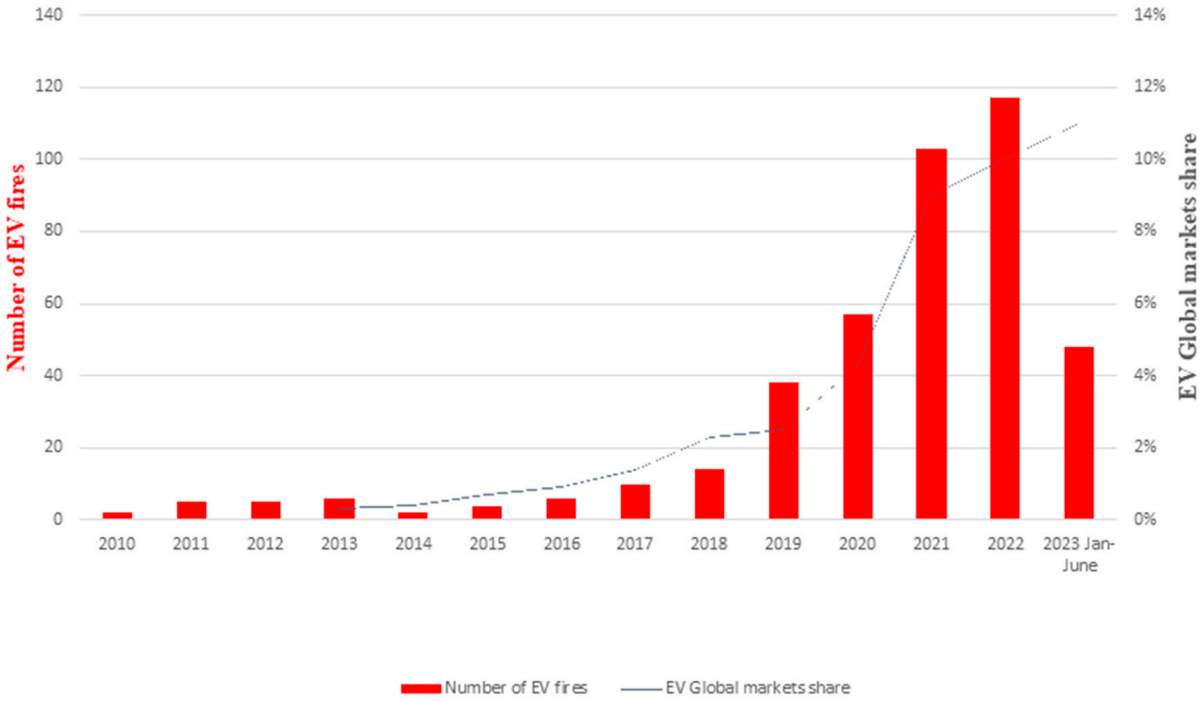


Figure 8. Number of light-duty EV fires by year and EV global market share [45].

Vehicle fires in underground parking present specific challenges, with electric vehicle fires introducing new and additional risks. Electric vehicles (EVs) pose unique challenges due to their high-voltage battery systems, which can lead to thermal runaway events and intense fires that are difficult to extinguish [46]. The thermal runaway of EV batteries can result in rapid and intense heat release, generating toxic gases and increasing the complexity of firefighting efforts [46]. In the same study, Trouchot highlights the evolution of car design, with new systems increasing the mass of plastic and new energy carriers, potentially affecting toxic gas emissions during fires, particularly in underground facilities like tunnels or car parks [46]. In an especially noteworthy finding, the study compares the emissions from electric vehicles (EVs) to those from traditional internal combustion engine cars. Contrary to some expectations, electric cars do not significantly increase the overall emission of toxic gases, although differences in specific emissions, such as hydrogen fluoride from batteries, were observed. These toxic gas emissions pose health and safety risks to occupants and emergency responders in underground parking facilities. The analysis of toxic gas emissions during vehicle fires is essential for developing effective smoke management strategies and ensuring the safety of individuals in enclosed spaces. This insight is vital for developing safety measures tailored to the growing number of EVs on the roads [46].

Another challenge is the reaction of EV batteries to external heat flux, which is a critical concern, especially in underground parking environments where fire spread and smoke movement can be challenging to control [47]. For example, the study of Tramoni [47] focuses on assessing the thermal impact on steel structures during fires caused by alternative fuel vehicles, including diesel, H2 fuel cells, natural gas, electric, and liquefied petroleum gas. During the tests, the highest temperature reached in different sections of the beams above cars ignited was higher than 900 °C, but the temperature reached by the column was colder than the beams and did not exceed 660 °C [47].

The emergence of EVs has raised concerns about fire hazards associated with battery failures and the potential for thermal events in underground parking structures. The reaction of high-pressure hydrogen storage vessels and lithium-ion batteries during fuel cell EV fires can pose risks to adjacent vehicles and structural elements in enclosed spaces like underground parking lots [48]. Understanding the fire load of new green-technology

vehicles, flame spread behavior, and thermal impacts on EV components is necessary for reducing fire risks in underground environments [48].

One critical aspect that could affect the accuracy of CFD simulations includes multimodal heat transfer. EV fires involve a combination of conduction, convection, and radiation, which can be difficult to model with high precision. The modeling of heat transfer in electric vehicle (EV) fires presents unique challenges due to the interplay of conduction, convection, and radiation. Each of these modes of heat transfer plays a critical role in the dynamics of fire spread and heat distribution, yet accurately simulating their combined effects in a Computational Fluid Dynamics (CFD) framework remains challenging. Convection is a significant mode of heat transfer in fire scenarios, particularly in the context of EV fires where the combustion of materials can generate significant airflow patterns. Maragkos and Beji highlight the importance of accurately modeling convective heat transfer in CFD simulations of fire-driven flows, noting that while local heat flux measurements in simpler scenarios can provide insights, they may not fully capture the complexities of real-world fires [49]. Radiation, while often considered a dominant mode of heat transfer in flames, may not be sufficient on its own to describe the heat transfer dynamics in EV fires. Recent findings indicate that relying solely on radiative heat transfer can lead to inaccuracies, as it does not account for the convective and conductive processes that also greatly influence fire behavior [50]. This is particularly relevant in the context of EV fires, where the combustion of battery materials can produce unique thermal profiles that necessitate a holistic approach to heat transfer modeling. The challenges of modeling conduction in the context of EV fires are underscored by the work of Florea et al., who discuss the complexities of heat conduction across different surfaces during fire scenarios [51]. The interaction between conductive heat transfer and the other modes can complicate thermal dynamics, particularly in the presence of materials with varying thermal properties, such as those found in electric vehicles. This complexity is further intensified by the need for accurate boundary conditions and material properties in CFD simulations. The integration of these heat transfer modes into a cohesive CFD model is critical for accurately predicting fire behavior. The study by Yan et al. illustrates the importance of coupled heat transfer models that consider interactions between different heat transfer mechanisms, which can lead to more accurate predictions of temperature distributions and fire dynamics [52]. This is particularly relevant for EV fires, where the simultaneous occurrence of combustion, turbulence, and heat transfer can create complex thermal environments that are challenging to simulate.

Another challenge in validating CFD models for EV fires is the lack of comprehensive experimental data that accurately reflects the unique combustion characteristics of lithium-ion batteries used in electric vehicles. Scala's study highlights the mechanical response of structural elements during EV fires, providing a finite element model to simulate heat transmission and structural reactions [53]. However, this model is limited by the absence of extensive experimental data on the thermal and combustion properties specific to EV fires, which hampers the ability to fully validate CFD predictions against real-world scenarios. The comparison made between internal combustion engine vehicles (ICEVs) and EVs in terms of fire risk underscores the need for more empirical data to inform CFD models effectively [53].

Several studies on the application of jet fans to smoke control and ventilation in tunnels exist [54–59]. However, just a few studies have been reported on the use of jet fan systems or impulse ventilation systems (IVSs) for smoke control or ventilation in covered car parks [12,21,26,60–64].

Ventilation systems, such as jet fans, are used for managing smoke in underground parking facilities, particularly during fire incidents. These systems are designed for carbon monoxide removal under normal conditions and smoke extraction during emergencies [27]. Jet fan ventilation systems are known to be effective in smoke extraction during emergencies, ensuring occupant safety and aiding firefighting efforts [58]. Studies have demonstrated that jet fans prevent smoke spread and enhance visibility for firefighters compared to

other ventilation systems [65]. Additionally, combining jet fans with ducting has been shown to expedite smoke removal, underscoring the importance of proper system design and configuration for effective smoke control [66]. Moreover, the use of jet fans can help maintain permissible CO concentrations in exhaust shafts, contributing to a safer environment in underground parking facilities [67].

Computational Fluid Dynamics (CFD) simulations are essential for optimizing ventilation systems, including jet fans, for smoke management in underground parking structures [43]. Researchers use CFD simulations to assess the effectiveness of these systems in controlling smoke dispersion during fire events, leading to improved design and placement of ventilation elements for enhanced safety [39]. Through CFD simulations, researchers can evaluate the performance of jet fans in controlling smoke dispersion during fire incidents, leading to enhanced design and placement of these ventilation components. The studies demonstrated that CFD simulations allow for the assessment of intricate interactions between fire-induced flows, smoke layering, and ventilation systems, offering valuable insights into the efficacy of jet fans in smoke control [26]. Moreover, CFD simulations enable the evaluation of various parameters, including fan configurations, ducting, and diffuser designs, to optimize the performance of jet fans in smoke extraction scenarios [68].

Additionally, CFD simulations aid in optimizing jet fan systems by analyzing factors like fan efficiency, airflow distribution, and pressure gradients within the ventilation network [69]. These simulations help visualize air streamlines, velocities, and pressure contours, facilitating the identification of optimal fan configurations for efficient smoke management in underground parking structures [70].

5. CFD Numerical Simulation Methodology

To optimize smoke removal in underground parking facilities through CFD simulations, a structured process is typically followed. This process includes model creation, scenario planning, and result interpretation. One of CFD's strength points lies in its adaptability—each simulation can be tailored to reflect the specific dynamics of an underground facility, from accounting for various ventilation designs to predicting how smoke and heat interact with structural elements. These simulations offer more than prediction—they guide engineers towards more informed decisions that increase safety and improve fire response strategies.

The initial step involves creating a digital model of the underground parking structure, incorporating features like vehicle lanes, parking spaces, ventilation systems, walls, pillars, and fire sources [18]. This phase entails defining the geometry, material properties, and boundary conditions required for simulating smoke dispersion during fire incidents [18].

Scenario planning encompasses setting simulation parameters such as fire location, intensity, and duration, as well as configuring the ventilation system, including jet fan placement and setup [12]. Additionally, airflow patterns, temperature gradients, and smoke layering are considered to replicate realistic fire scenarios and evaluate smoke management strategies [71]. This study made by Santoso et al. investigates the impact of ventilation systems on smoke spread in underground car parks during fire incidents. Their research utilized the Fire Dynamics Simulator (FDS) version 6 to model various configurations and observed how factors such as ceiling beams and building structures influenced smoke flow patterns. The CFD results were validated against experimental data, demonstrating that the model could accurately predict smoke behavior under different conditions, which is crucial for effective fire safety design in underground facilities [71]. This finding aligns with the work of Fuh and Wei [72], who conducted a numerical study on mechanical ventilation in underground tunnels. Their simulations were compared with full-scale experimental data, revealing that CFD could effectively predict temperature distributions and visibility during fire events, thus confirming its reliability for tunnel fire safety assessments [72].

Following the simulation runs, result interpretation is necessary for assessing the ventilation system's performance in smoke removal. Researchers analyze data on smoke concentration, airflow velocities, temperature distributions, and visibility profiles to evalu-

ate the system's efficiency in evacuating smoke from the parking structure [26]. Comparing simulation outcomes with safety standards and performance criteria aids in identifying areas for enhancement and optimizing the ventilation system for improved smoke control [21]. This research highlighted the complexity of smoke spread dynamics and the necessity of using CFD to visualize fluid flow and compare design alternatives, thereby minimizing the risk of miscalculations that could arise from conventional methods [21].

Conducting CFD simulations for underground parking smoke removal involves a meticulous approach to model creation, scenario planning, and result interpretation. This method allows researchers to evaluate ventilation system effectiveness, particularly jet fans, in managing smoke during fire incidents and implement strategies to enhance safety in underground parking facilities.

In conclusion, the comparison of CFD results with experimental data in the context of fire dynamics in underground buildings is essential for validating and refining fire models. The studies reviewed illustrate the successful application of CFD in various underground fire scenarios, highlighting its predictive capabilities and the importance of experimental validation in enhancing model accuracy. Continued research in this area will further improve the reliability of CFD as a tool for understanding and mitigating fire hazards in underground environments.

6. Case Studies

The focus of the study made by Çakir and Ün [63] is the design and simulation of a jet fan system for smoke and temperature control in the car park area of a hospital using CFD. The car park is 19,438.59 m^2 and was divided into five zones for the CFD smoke analysis. The study involves creating various fire scenarios in line with international standards to determine the optimal placement of jet fans and to assess the system's efficiency in smoke and temperature control. They modeled the car park, using Ansys 18.2 software for the CFD analysis, and considering several factors like temperature, gas, smoke concentration, air velocities, and the effect of obstacles in the car park. CFD simulations allowed the authors to precisely model air velocity, smoke distribution, and temperature gradients in the car park. The authors employed a rectilinear mesh with cells ranging from 0.1 to 0.35 m, ensuring a high degree of spatial accuracy. With over 5 million mesh cells used, the analysis could account for complex geometries and obstacles, which is crucial for simulating airflow in real-world environments. The simulations track how smoke disperses from the fire location and how quickly the jet fans can clear it. The simulation showed that the temperatures in the car park remained below 60 °C, and visibility was always more than 5 m, which is crucial for safety. The study highlighted the importance of the jet fan's placement and the balance between air speeds and exhaust rates to ensure efficient smoke removal. The CFD simulations enabled the researchers to validate their design choices, particularly regarding the number of jet fans, their speed, and placement within the parking zones. The report includes multiple figures illustrating velocity contours, smoke concentration levels, and temperature profiles, providing comprehensive evidence for the system's efficiency. The authors emphasize that CFD simulations are invaluable for refining jet fan system designs before implementation, reducing the likelihood of errors in real-world deployment.

The study concludes that CFD analysis is vital for designing effective jet fan systems for smoke and temperature control in enclosed parking areas. Such findings likely influence building codes and standards by emphasizing the need for CFD analysis in the design phase of underground parking ventilation systems, ensuring temperatures remain below critical thresholds, and visibility is maintained during a fire event. It emphasizes the need for detailed modeling and simulations to determine optimal fan capacities and placements for effective smoke management [63].

In the study made by Balan et al. [64], they focused on evaluating the efficacy of a ventilation system equipped with impulse fans in underground parking during fire incidents. They analyze a ventilation system with impulse fans in an underground car park to ensure optimal air parameters and evacuate smoke during a fire. The case study involved

the ventilation system of an underground parking lot with a capacity for 250 vehicles. The study area had a total surface of 6750 m^2 and a volume of 23,625 m^3, which represented a real-world scenario for testing. Various operating stages of the ventilation system were analyzed to determine the system's ability to control carbon monoxide (CO) concentration and smoke distribution during a fire.

The ventilation system is primarily analyzed for two main functions. First, its normal operation focuses on maintaining air quality and comfort by effectively reducing CO concentration. Secondly, its emergency operation is crucial for smoke evacuation in the event of a fire. To evaluate the performance of the ventilation system under different conditions, Computational Fluid Dynamics (CFD) simulations were conducted. These simulations covered four distinct scenarios: the normal operation of the ventilation system, the combined operation of the ventilation system and impulse fans for regular use, emergency operation during a fire using only the ventilation system, and a combined operation during a fire incorporating both the ventilation system and impulse fans. The result was that in normal operation without impulse fans, CO concentration exceeded safe levels (360 PPM vs. 100 PPM allowed), posing a risk to users. Adding impulse fans reduced CO concentration to safer levels (60 PPM).

In fire scenarios, areas of high smoke concentration and poor visibility were identified without impulse fans, hindering emergency response and evacuation. Incorporating impulse fans significantly improved air circulation, reduced smoke concentration, and enhanced safety and visibility; this could lead to regulations requiring the integration of impulse fans with conventional ventilation systems in underground parking. The integration of CFD simulations played an important role in this study [64], offering a detailed visualization of air movement, pollutant dispersion, and the impact of design modifications. The use of CFD enabled the researchers to simulate different scenarios and evaluate the performance of the ventilation system both in normal and emergency conditions. The CFD simulations provided crucial insights into how the system could be optimized. By identifying stagnation zones, the research team was able to propose the use of impulse fans in problematic areas. These fans improved air circulation and directed polluted air and smoke towards exhaust outlets more effectively. Furthermore, the simulations facilitated precise adjustments to the system, allowing it to meet safety regulations for both CO levels and smoke evacuation during fires. Without the detailed data provided by the CFD analysis, such refinements might have been missed, potentially resulting in a system that did not adequately protect the building's occupants. The integration of CFD simulations with the case analysis provided a comprehensive understanding of the ventilation system's strengths and weaknesses. The findings demonstrate that while traditional methods might suffice for smaller, simpler environments, complex spaces like underground parking lots require advanced simulation techniques to ensure safety and efficiency. CFD was not only instrumental in diagnosing the system's issues but also in optimizing it to meet safety standards under both normal and emergency conditions. The study concludes that the integration of impulse fans with the ventilation system improves air quality under normal conditions and enhances safety and effectiveness in smoke evacuation during fires. The optimal positioning and number of impulse fans are crucial for the system's efficiency [64].

The determination of heat release rate (HRR) in vehicle fires has been a focal point in past experimental studies by Mangs and Keski-Rahkonen (1994), Shipp and Spearpoint (1995), and Cheng and John (2002). These studies have provided valuable insights into the behavior of vehicle fires and the associated HRR. Furthermore, investigations in enclosed-basement fires have focused on smoke movement, emphasizing the significance of understanding smoke dynamics in such environments [73]. The study concluded that the fastest smoke-removal time in a basement fire was achieved with a combination of sprinklers, make-up air fans, smoke-extraction fans, jet fans, and ductwork. The insights from these studies could inform standards related to the required capacity and design of ventilation systems in underground parking to address the high HRR values of modern vehicle fires.

The full-scale experiments conducted by [74] have revealed the substantial heat release rate (HRR) values associated with modern car fires, particularly exceeding 16 MW when three cars are involved in the fire incident. The document provides a thorough investigation into smoke movement in car park fires, integrating both full-scale and reduced-scale testing alongside Computational Fluid Dynamics (CFD) simulations to enhance fire safety strategies. A car park was investigated composed of a simplified layout like a wide road tunnel with a plain ceiling and walls, and no consideration for horizontal and vertical beams supporting the ceiling. The research employs both full- and reduced-scale models to simulate fire scenarios and study smoke behavior in simplified car park geometries. Full-scale tests were conducted in a 28.6 m × 30 m × 2.6 m car park model using a hexane pool burner to replicate varying heat release rates (200 kW to 4000 kW). Simultaneously, reduced-scale experiments were carried out with geometrical, velocity, and temperature scaling to mimic real-world conditions. The reduced-scale model employed helium and air mixtures to replicate buoyancy effects, and Particle Image Velocimetry (PIV) was used for velocity measurements. CFD simulations from Ghent University complement the physical tests by providing additional insights into the air and smoke flow patterns, as well as the behavior under different ventilation rates. These simulations helped identify critical phenomena, such as the formation of recirculation bubbles near entrances and stratification near extractors, which were not easily observable in full-scale tests alone. Reduced-scale tests and CFD simulations uncovered unexpected recirculation zones near the car park entrances and cooler temperatures at extractors due to fresh air drafts. These findings highlighted the importance of CFD in diagnosing complex flow structures that may not be directly observable during physical tests. By comparing the experimental data to CFD simulations, empirical formulas were developed to predict smoke behavior in car parks with flat ceilings. The study also revealed that traditional tunnel fire models do not entirely capture the nuances of car park fires, thus requiring adjustments or new correlations, such as those proposed by Kunsch for critical velocities. Reduced-scale models were validated against full-scale experiments using CFD simulations, confirming their reliability for investigating more complex car park geometries in future studies. This validation is particularly important for scenarios that are expensive or impractical to recreate in full-scale settings. This makes the results specifically applicable to large, closed parks with flat ceilings and unidirectional smoke and ventilation patterns. A crucial aspect managed by a ventilation system is the air's longitudinal velocity, particularly in the event of a fire. In cases where the system is not optimally designed, smoke from the fire can reach the ceiling, creating two streams of gas and causing the smoke to spread throughout the parking, known as the back layering phenomenon. An important outcome of the study is the determination of critical velocity, the velocity at which no smoke occurs upstream in the ventilation flow relative to the fire source. This is crucial for ensuring that smoke does not impede firefighting efforts. In conclusion, ref. [74] study provides valuable insights into the dynamics of smoke movement in car park fires, emphasizing the importance of understanding smoke back layering and critical velocity for effective fire safety management. The results are particularly relevant for large, closed car parks with simple geometries, offering empirical formulae and models for predicting smoke behaviors in these environments.

The study illustrates that CFD simulations are a powerful tool for interpreting complex flow dynamics in smoke ventilation systems, especially when physical testing is limited by scale or cost. The integration of CFD with full- and reduced-scale experiments strengthens the reliability of results and helps formulate practical guidelines for designing smoke control systems in car parks. The study provides both experimental validation and numerical modeling insights, ensuring that future designs are both cost-effective and scientifically grounded.

The importance of this research is particularly relevant in the context of fire safety and risk management in enclosed environments, where the potential for rapid fire growth and high heat release rates necessitates robust fire ventilation systems. Understanding the

dynamics of fire incidents involving modern vehicles is crucial for developing effective fire safety measures, including the optimization of ventilation systems in car parks and other enclosed spaces. Furthermore, the integration of Computational Fluid Dynamics (CFD) simulations and experimental data can provide valuable insights into the behavior of fires with high HRR values, aiding in the development of advanced fire ventilation strategies and systems [54,57].

Additionally, studies on the effects of fuel type, fuel distribution, and vent size in under-ventilated compartment fires have contributed to understanding fire behavior in enclosed spaces made by [75]. The studies have investigated the scaling factors for different fires, demonstrating the importance of considering under-ventilated scaling relationships in fire scenarios. This research on under-ventilated fires in a room with varying vent sizes and polymer fuel types has provided valuable data on local equivalence ratio and toxic gas species, contributing to a deeper understanding of fire dynamics. The inclusion of soot into mixture fraction analysis has allowed for the identification of fuel-rich or under-ventilated conditions in compartment fires of smoky fuels, further enhancing the understanding of fire behavior.

The location of the jet fans in underground parking can significantly affect their performance. The positioning of a fan near a wall or ceiling can lead to pressure drops, which in turn influence how effectively the fan transfers energy to the induced airflow.

The design of the blades in a fan's impeller can impart a rotational component to the airflow, resulting in a helical motion, but the air jet shape is also affected by the presence of silencers or nozzles at the fan's outlet. These components can modify the shape and direction of the air jet emitted by the fan.

The performance of both single and twin jet fan installations in conjunction with conventional ventilation systems was study Sittisak [76]. The study focuses on enhancing ventilation efficiency in underground car parks to remove harmful gases like carbon monoxide (CO). It examined various parameters like operation mode, tilt angle, and gap distance of jet fans. It was also used CFD to simulate different scenarios for jet fan installations. The results indicate that jet fans, especially when inclined, can enhance ventilation performance. Twin jet fans showed a stronger ability to move and disperse air compared to single jet fans and it was established that inclined single jet fans offer a balance between ventilation efficiency and energy utilization.

The integration of CFD simulations into the study provided deep insights into airflow dynamics and pollutant dispersion in a complex underground environment. By simulating various configurations, the study was able to identify key performance trade-offs between different jet fan designs, offering actionable recommendations for enhancing ventilation efficiency in underground car parks. This expanded analysis shows not only the practical applicability of the study's findings but also emphasizes the rigorous computational and experimental methods used to validate and refine the proposed ventilation solutions during fire scenarios using CFD simulations.

Another important characteristic is the shape of the air jet depends also on the presence of silencers or nozzles on fan delivery (Figure 9). Experimental studies made by Martegani and Pavesi (1994) examined how the air jet from a single fan positioned on the tunnel ceiling behaves. They conclude that the jet stays close to the ceiling for a distance of at least 40 fan diameters [77]. In the event of an underground parking fire, an important factor for ensuring safety is managing back-layering smoke [77]. This occurs when fire-induced smoke reaches the car park's ceiling, creating two upward-moving gas streams. To prevent this, a well-designed ventilation system is necessary, capable of achieving critical ventilation velocity. This velocity is defined as the minimum required to prevent the smoke from rising [77].



Figure 9. Axial Jet Fan installed in an underground parking car.

Performances of several configurations of the ventilation system during fire scenarios using CFD simulations have been studied by (Li and Chow, 2003) [78]. The systems reviewed include longitudinal, semi-transverse, transverse, partial transverse, and combined ventilation systems. Each system's performance is assessed by simulating airflow and smoke propagation, key determinants in ensuring passenger safety during fires. They have highlighted the advantages and disadvantages of each of them, as reported in the study [78]. CFD is leveraged as a powerful tool to simulate and analyze complex fire dynamics in tunnels, which can be difficult to replicate in full-scale experimental settings due to high costs and logistical challenges. Through CFD, the authors examine variables like air velocity, temperature distribution, and smoke movement to assess the efficiency of the ventilation systems under fire conditions. A tunnel ventilation system installed longitudinally has the advantages of requiring less space for ventilation building and ductwork and less initial cost, but the application is limited by tunnel length, and the smoke is not removed from the tunnel. In case the tunnel system ventilation is installed transversely, the solution is suitable for long tunnels and is applicable for bi-directional traffic, but the disadvantages are large ventilation buildings and ductwork and high investment costs [78]. The combination of the longitudinal and semi-transverse system provides good smoke control, and the smoke is removed from the tunnel, but this comes with high investment costs and higher operating and maintenance costs. The use of CFD in this study provided an in-depth analysis of various tunnel ventilation systems under fire conditions, offering valuable data on airflow, smoke movement, and temperature distribution. Through comprehensive numerical simulations, the study emphasizes the importance of fine-tuning ventilation systems to enhance evacuation safety and fire response measures. CFD stands out as an essential tool in optimizing tunnel safety design, especially when physical testing is impractical or prohibitively expensive.

The study made by Lulea [17] presents an important study focused on developing and validating a Computational Fluid Dynamics (CFD) model for simulating fire dynamics in indoor environments. It studied the interaction between fire protection systems, specifically sprinkler and ventilation systems, in various fire scenarios. It was developed a CFD model that can simulate the indoor environment during a fire, particularly focusing on air temperature near sprinkler systems and how this interacts with ventilation systems. The researchers created real-scale experiments, and the data were used for a CFD model with FDS soft. The integration of CFD with the case study allowed for a detailed analysis of fire dynamics, enabling the researchers to predict how the interaction between the sprinkler system and the ventilation system affects fire development. This integration is key in exploring the behavior of critical parameters such as the heat release rate (HRR), temperature

distribution, and visibility under different fire scenarios. These experiments involved a controlled setup using burners, ventilation systems, and various sensors to measure indoor air temperature, visibility, oxygen, and carbon dioxide concentrations. Different ventilation airflow rates were tested to assess their impact on the indoor environment during a fire. The conclusions were that increasing ventilation rates lead to lower indoor temperatures near the sprinkler, potentially delaying or preventing sprinkler activation, and ventilation impacts vary across different areas of the room, indicating a complex interaction between air movement and temperature distribution. This study outlines the robust integration of CFD simulations with case analysis in the document, emphasizing the depth of the fire dynamics investigation and its relevance to fire safety system design. The study's successful use of CFD to simulate real-world fire scenarios demonstrates the critical role of such tools in modern fire safety engineering.

Kmecová [21] designed and simulated two fire ventilation system alternatives for an underground car park using CFD: Alternative A1: Exhaust shafts in both parts of the car park with jet fans and an air exchange rate of 10 times per hour. Alternative A2: Exhaust shafts on one side, opposite the main air supply, with an air exchange rate of 15 times per hour. The two alternatives were rigorously compared through CFD simulations, which modeled fire dynamics, fluid flow, and combustion products, offering invaluable insights into the interaction between fire-generated smoke and ventilation systems. Alternative A1 was less effective, with inadequate ventilation and low visibility of 600 s after extinguishing the fire. In contrast, Alternative A2 completely exhausted the smoke and significantly improved visibility, ensuring a safer environment for evacuation. Thus, Alternative A2, with its higher air exchange rate and single-sided exhaust shafts, was more efficient in smoke removal and visibility enhancement during fire emergencies [21]. The research concluded that the design and positioning of ventilation elements (like exhaust shafts and jet fans) are critical for the effectiveness of fire ventilation systems in underground car parks. Building codes may adapt to include specifications on the air exchange rate, positioning of exhaust shafts, and the use of inclined jet fans to optimize smoke removal and improve safety during fire emergencies.

The integration of CFD simulations provided a robust framework for analyzing complex fire dynamics, offering valuable insights into the practical considerations of designing safe and efficient ventilation systems for underground car parks. The results emphasized the critical role of proper fan and exhaust shaft placement in ensuring the safety of occupants and firefighters, highlighting the limitations of relying solely on air exchange rates without considering system layout.

Overall, the report reinforces that CFD simulations are indispensable for designing fire ventilation systems, especially in environments with complex geometries like underground car parks, where miscalculations could lead to life-threatening consequences.

The critical ventilation velocity was studied by Nele Tilley and Bart Merci [24]. This study presents a detailed computational analysis investigating the relationship between ventilation velocity and smoke back layering distance in large, closed car parks. Utilizing over 350 Computational Fluid Dynamics (CFD) simulations, the research delves into various parameters influencing smoke behavior and ventilation efficiency during fire incidents. The approach incorporates both theoretical insights and numerical experiments to formulate analytical relationships that can aid in designing effective smoke control systems in enclosed spaces, specifically car parks. It was observed that the critical ventilation velocity increases with the fire source area and heat release rate per unit area, and with an increase in car park height. A slight decrease was noted with increasing the car park width. There is a linear relationship between back layering distance and the difference between critical and actual ventilation velocities. The coefficient in this relation is independent of the fire source area, car park height, and width but increases with decreasing heat release rate per unit area [24]. Fire safety protocols might incorporate guidelines for achieving this critical velocity through the strategic placement and operation of ventilation equipment, ensuring that smoke does not impede firefighting efforts or evacuation paths. The use of CFD simulations

is pivotal to the study. CFD allows for the modeling of complex airflow and smoke behavior that would be impractical to measure in full-scale real-world tests. The study employs a Fire Dynamics Simulator (FDS), a sophisticated CFD tool, to simulate smoke spread and ventilation effectiveness in varied car park setups. The study also emphasizes the challenges presented by non-uni-directional flow patterns, such as those caused by partial obstructions at car park entrances. In some cases, complex recirculation regions can form, significantly altering the smoke back layering behavior. The CFD simulations illustrate these effects, providing detailed visualizations of flow patterns under different configurations. The findings highlight the importance of ensuring that car park ventilation systems are designed to maintain uni-directional flow patterns, as deviations can lead to unpredictable smoke behavior and reduced system effectiveness. This CFD-based study offers a thorough exploration of the relationship between ventilation velocity and smoke back layering distance in large car parks. Through detailed parameter variation and integration of over 350 simulations, the researchers have provided a set of practical formulae that can guide the design of effective smoke and heat control systems. The integration of CFD simulations with full-scale experimental validation underscores the reliability of the findings, making this research a valuable contribution to fire safety engineering.

These findings advocate for the integration of advanced simulation tools and experimental data in the development of fire safety strategies for underground parking. Regulatory bodies might update or introduce new standards that require the implementation of dual-function ventilation systems, specify guidelines for the design and positioning of ventilation elements, and recommend practices for achieving critical ventilation velocities. The emphasis on understanding fire dynamics, including the behavior of smoke and heat in fire scenarios, suggests a trend towards more sophisticated and scientifically informed approaches to fire safety in underground parking design and management.

In conclusion, the alignment between smoke diffusion characteristics observed in real-world scenarios and CFD model simulations is highly evident across multiple case studies. The CFD models, through their ability to precisely simulate airflow, temperature gradients, and smoke distribution, provided valuable insights into the effectiveness of different fire ventilation and smoke management strategies. In each case, CFD simulations allowed for the optimization of jet fan systems, impulse fans, and overall ventilation designs, ensuring the systems could efficiently handle smoke diffusion in complex environments like underground car parks.

Furthermore, CFD simulations were instrumental in identifying key issues, such as back-layering and stagnation zones, which are critical in fire safety. These simulations were able to replicate these dangerous phenomena and suggest solutions, such as achieving critical ventilation velocities and optimal fan placements, that directly impact real-world fire safety strategies. The integration of CFD modeling into these case studies demonstrates its essential role in both the design and validation phases of fire ventilation systems, making it a powerful tool for enhancing safety in enclosed environments prone to smoke diffusion during fires.

This strong alignment between the observed smoke diffusion behaviors and CFD simulation outcomes underscores the importance of leveraging computational modeling. It helps designers and engineers refine ventilation strategies, ensuring systems are not only effective but also comply with safety standards. This confluence of CFD simulations and real-world observations advocates for their continued use in developing advanced, scientifically grounded fire safety systems, particularly in environments with complex geometries such as underground car parks. Future regulatory frameworks and building codes will likely be influenced by these findings, emphasizing the need for CFD-based analysis in fire safety design.

The case studies underscore the importance of considering fire size and spread when designing ventilation systems. For example, in scenarios where fire size was underestimated, the ventilation systems failed to maintain visibility and prevent back-layering, leading to hazardous conditions for evacuees. By integrating fire size dynamics into CFD

models, a more accurate prediction of fire spread and smoke behavior was achieved, leading to improved fire safety measures

7. Challenges and Limitations of CFD

The current challenges of Computational Fluid Dynamics (CFD) simulations in optimizing smoke removal from parked cars involve several key aspects, including computational requirements, model accuracy, and the need for empirical validation.

One of the primary limitations of CFD simulations is their sensitivity to input data, such as the heat release rate and ventilation parameters. Experimental tests, by contrast, offer direct measurements that reflect real-world fire dynamics. Discrepancies between CFD results and experimental data often arise when models do not fully capture complex phenomena like back-layering or convective heat transfer. Addressing these gaps is a key challenge in improving CFD accuracy.

Another critical limitation is the computational demand associated with conducting CFD simulations for smoke removal strategies in parking structures, as highlighted by Nakamura and Hajjawi [18]. The complexity of modeling smoke dispersion in confined spaces like underground parking lots requires significant computational resources, which can pose challenges in achieving timely and cost-effective simulations.

Model accuracy is another crucial limitation, as discussed by Viegas [79], emphasizing the importance of validating CFD models for smoke control in underground car parks. Ensuring that the CFD models accurately represent the real-world behavior of smoke dispersion and ventilation systems is essential for reliable simulation outcomes and effective smoke management strategies. Factors such as the resolution of the model, boundary conditions, and turbulence modeling can influence the accuracy of CFD results and the effectiveness of smoke removal strategies.

Empirical validation is a fundamental requirement for enhancing the credibility and applicability of CFD simulations in smoke removal from parked cars, as emphasized by Santoso [71]. Conducting validation studies against real-scale tests and experimental data are essential to verify the accuracy and reliability of CFD models, particularly in complex environments like underground parking facilities.

Addressing the limitations of CFD simulations in optimizing smoke removal strategies for parked cars necessitates overcoming challenges related to computational requirements, ensuring model accuracy, and conducting empirical validation studies to enhance the reliability and effectiveness of simulation outcomes.

Simulating electric vehicle fires in CFD encounters significant challenges due to the unknown risk profile and fire characteristics associated with lithium-ion batteries. One of the critical issues is the combustion behavior of lithium-ion batteries, which differs significantly from traditional internal combustion engine fires. The unique challenges posed by battery combustion, such as rapid thermal runaway, the release of highly toxic gases, and the potential for re-ignition, necessitate the development of highly realistic models to accurately simulate these scenarios. Current CFD simulations must evolve to incorporate detailed physical and chemical processes specific to battery fires to ensure that safety strategies are both effective and reliable. This advancement is needed not only for enhancing the predictive accuracy of these models but also for optimizing the design of ventilation systems and emergency response protocols in environments increasingly populated by EVs. Without such realistic and validated models, the risk of inadequate smoke management and potential safety hazards remains high, underscoring the importance of continued research and development in this critical area. The first limitation is the understanding of fire characteristics when accidents involving lithium-ion batteries (LIBs) in electric vehicles are prevalent globally, yet the internal defects leading to thermal runaway and fire incidents are challenging to observe and replicate [80]. The lack of detailed knowledge about the fire characteristics of modern electric cars hampers the accurate simulation and prediction of fire events.

There is a scarcity of comprehensive data on the ignition, combustion, and smoke generation characteristics of electric vehicle fires [80]. This lack of data impedes the development of precise simulation models for optimizing smoke removal strategies in underground parking facilities during electric vehicle fires.

The complexity of battery fires lithium-ion battery fires in electric vehicles present specific challenges due to the thermal runaway phenomenon [81]. Factors such as short circuits, overcharging, and elevated temperatures can trigger thermal runaway reactions, leading to the release of flammable electrolytes and making fire suppression challenging.

Empirical validation is therefore necessary to verify the reliability of CFD simulations in predicting fire dynamics and optimizing smoke removal strategies during electric vehicle fires [82]. Experimental data on electric vehicle fires is required to validate the accuracy of simulation models and ensure their applicability in real-world scenarios.

The challenges of simulating electric vehicle fires in CFD stem from the intricate nature of lithium-ion battery fires, the limited understanding of fire characteristics, the scarcity of data for accurate simulations, and the critical need for empirical validation to enhance the reliability of simulation outcomes in underground parking facilities. Modern technology provides CFD-based models with greater computing power so they can model each phase of the fire from the ignition, growth, to the flashover and decay [83].

When examining the challenges and limitations of utilizing Computational Fluid Dynamics (CFD) in smoke evacuation from electric fires in parking facilities, several crucial aspects emerge. The gaps in understanding fire dynamics and smoke movement in large spaces, as highlighted in [73], present a challenge in accurately modeling smoke evacuation scenarios. This article [73] explores the dynamics of fire and smoke in large spaces, specifically in a 20 m cubic atrium, using pool fires of 1.3 and 2.3 MW. It focuses on the performance-based design for fire safety in modern buildings, like high rises and transport stations, which often feature large open spaces such as atria. This research is valuable because knowledge gaps still exist regarding fire dynamics and smoke movement in such large volumes.

The study reported on the Murcia Atrium Fire Tests, which involved detailed measurements of gas and wall temperatures, airflow, and exhaust fan pressure drops. These measurements were then compared against predictions made by the Fire Dynamics Simulator (FDS) Computational Fluid Dynamics model. The tests aimed to validate CFD modeling techniques used in fire safety design, ensuring they are reliable for predicting real-world scenarios.

Overall, the comparison between experimental results and CFD simulations showed good agreement, particularly in the far field of the plume, although there were some inaccuracies in the lower plume region and near the flame. This research contributes valuable experimental data to the field, supporting the development of more accurate predictive models for fire safety engineering [73].

Additionally, the study of tunnel fires and the impact of longitudinal ventilation on smoke management, as investigated by Khattri [84], provides an understanding of the complexities of simulating smoke evacuation in enclosed environments. The article explores the impact of varying oxygen concentrations in ventilating air on the dynamics of tunnel fires. It highlights the use of Computational Fluid Dynamics (CFD) simulations to study how mixing inert gases like nitrogen or carbon dioxide with ambient air to lower oxygen levels can influence critical safety parameters such as maximum tunnel ceiling temperature, fire growth rate, and maximum heat release rate. The study utilizes Froude scaling to extrapolate findings from reduced-scale models to real tunnel scenarios, ensuring that the simulations reflect realistic fire behaviors. Results indicate that adjusting oxygen concentration in ventilation air can significantly affect fire dynamics, potentially enhancing safety by controlling the maximum heat release rate and providing safer conditions for evacuation and firefighting. The research contributes valuable insights for designing ventilation systems in tunnels, focusing on optimizing oxygen concentration to manage fire risks effectively while ensuring safe environments during emergencies [84].

Other significant perspectives on using Computational Fluid Dynamics (CFD) for fire safety evaluations, especially regarding smoke evacuation in parking areas, including those with electric fires, was studied by Brzezińska and Bryant [85]. Technical solutions were analyzed for mitigating fire hazards from electric vehicles and methods were presented of performance-based analysis for assessing fire safety in car parks. The focus was on understanding and mitigating these risks through performance-based analysis and using Computational Fluid Dynamics (CFD) simulations to model fire scenarios. The use of the Fire Dynamic Simulator (FDS) software for predicting smoke dispersion and temperature distribution during an EV fire is highlighted, illustrating its utility in planning for safe evacuation and firefighting operations [85].

The challenges and limitations of using Computational Fluid Dynamics (CFD) in smoke evacuation for semi-open car parks, as detailed in the study made by Heijden [80], revolve primarily around the complexity of accurately modeling fire dynamics and smoke behavior in these structures. According to the study, the CFD models need to be validated against empirical data or other reliable simulations to ensure accuracy. This is often challenging due to the unique conditions present in each scenario, such as varying ventilation conditions and the geometry of the car parks. The primary hurdle lies in accurately modeling the complex fire dynamics and smoke behavior within these structures. Validating the CFD models using Ansys Fluent against real-world data is crucial, but this is often difficult due to the unique ventilation systems and geometries of each car park. Additionally, the standard modeling approach used the k-ε model (Reynolds Averaged Navier–Stokes method), which might not capture all the intricacies of turbulent smoke movement and heat transfer. This could limit the model's ability to predict smoke behavior in scenarios with greater fire severity or different conditions.

Another significant challenge is incorporating the unpredictable nature of wind. Wind speed and direction significantly impact smoke and fire spread, influencing natural ventilation effectiveness. Accurately modeling wind effects is necessary and challenging due to its constant variations. Furthermore, the study highlights the impact of car park geometry and structural variations on smoke and heat flow. Beams, ceiling heights, and the open façade area can all influence smoke paths. This complexity necessitates specific modeling approaches or adjustments for each individual car park layout.

The study also emphasizes the importance of using realistic fire scenarios in simulations. Assumptions regarding the number of burning cars and fire progression can significantly influence the results. Real-world fires often involve multiple vehicles burning simultaneously with varying fire intensities, which may not be fully captured by current simulations.

The research also raises concerns about the adequacy of current safety standards. Even car parks designed according to existing guidelines might not always meet safety criteria for smoke evacuation. This suggests potential deficiencies in both the guidelines and the models used to enforce them [86].

7.1. Enhanced Precision in Smoke Behavior Numerical Modeling

Recent developments in CFD technology have led to enhanced precision in numerical modeling of smoke behavior. This improvement is due to better algorithms that can simulate the intricate interactions between smoke, fire, and the surrounding environment with higher accuracy. Such precision allows for the design of more effective smoke removal systems by predicting smoke movement patterns more reliably, enabling targeted intervention in critical areas. To improve precision in smoke behavior modeling using Computational Fluid Dynamics (CFD), a comprehensive approach can be implemented drawing from numerous studies [87], which highlighted the effectiveness of CFD in predicting safety parameters related to smoke toxicity and visibility in combustion processes, highlighting the potential for accurate modeling of smoke behavior [87]. The study made by Gunjal [88] emphasized the versatility of CFD in studying complex flow phenomena, such as single-phase flow in packed beds, which can be valuable in smoke modeling [88].

Moreover, Sun [89] demonstrated the practical application of CFD in fire safety scenarios by interpreting flow fields induced by water spray systems in tunnel fire experiments [89]. Additionally, Zhu [90] stressed the significance of CFD models based on fundamental conservation principles for evaluating flow and reactions in various reactor types, providing a solid foundation for smoke behavior modeling using CFD [90]. By synthesizing insights from these studies, a comprehensive smoke behavior model employing CFD can encompass factors like smoke toxicity, visibility, flow dynamics in packed beds, fire safety scenarios, and reactor behavior. This integrated approach can lead to a more accurate understanding and prediction of smoke behavior, thereby improving safety measures and decision-making in fire-related situations.

7.2. Integration with Artificial Intelligence and Machine Learning

To improve the precision in smoke behavior modeling from fires, integrating Computational Fluid Dynamics (CFD) with Artificial Intelligence (AI) and Machine Learning (ML) has been proposed as a promising approach. By combining the capabilities of CFD for fluid dynamics simulations with the data processing and predictive power of AI and ML, a more comprehensive and accurate smoke behavior model can be developed.

Studies by Brunton [91] and Pathak [92] highlight the potential of machine learning in fluid mechanics and chaotic systems prediction, respectively. These insights can be utilized to develop AI and ML algorithms that can analyze the complex data generated by CFD simulations of smoke behavior from fires. Additionally, the works by Pishnamazi [93] and Babanezhad [94] demonstrate the application of AI, specifically the Adaptive Neuro Fuzzy Inference System (ANFIS), in conjunction with CFD for fluid flow predictions, which can be adapted for smoke dispersion modeling.

Furthermore, the study by Ermolieva [95] discusses the integration of AI and machine learning for decision support systems, emphasizing the iterative learning procedures that can refine smoke behavior models. Moreover, Vinuesa [96] suggests enhancing CFD with machine learning to improve turbulence closure modeling, which is crucial for accurately simulating smoke dispersion patterns.

By synthesizing these studies, a novel approach can be developed that integrates CFD simulations of smoke behavior with AI and ML algorithms to predict smoke dispersion, toxicity, and visibility in fire scenarios. This integration can lead to more precise and efficient smoke behavior modeling, aiding in decision-making processes and enhancing safety measures during fire incidents.

7.3. Real-Time Simulation and Decision Support

Real-time Computational Fluid Dynamics (CFD) simulations are enhancing the capabilities of emergency responders in handling fire incidents. The integration of advanced computing power and sophisticated CFD software allows for real-time analysis of fire dynamics, providing crucial decision support for emergency responders. By predicting the spread of smoke and fire behavior, real-time CFD numerical simulations enable responders to efficiently plan the deployment of smoke removal techniques and evacuation procedures, potentially leading to saving lives and reducing property damage.

Maragkos and Merci [97] conducted a study on Large Eddy Simulations of CH4 fire plumes, focusing on understanding the dynamics and oscillatory behavior of large-scale fire plumes. This research evaluates the predictive capabilities of turbulence and combustion models, which are essential for improving fire incident simulations [97]. Additionally, Fernánez-Alaiz [98] analyzed fire propagation in a sublevel coal mine using experimental analysis, CFD modeling, and simulations, demonstrating the practical application of CFD in comprehending fire dynamics in complex environments [98]. The importance of CFD simulations in advancing fire safety science is emphasized by [49], highlighting the role of CFD in modeling multi-physics phenomena such as turbulence, combustion, and heat transfer in fire-driven flows. Furthermore, Alós-Moya [99] displayed the analysis of a bridge failure due to fire using CFD and finite element models, emphasizing the signifi-

cance of CFD in understanding the thermomechanical response of structures under fire conditions [99].

Real-time CFD simulations, as discussed by [100], have the potential to replicate fire behavior observed in large compartments, aiding in the development of effective firefighting strategies. Moreover, Caliendo [101] introduced a risk analysis model for road tunnels using CFD, underscoring the importance of CFD in evaluating the impact of natural ventilation on user safety during tunnel evacuations.

In conclusion, the integration of real-time CFD simulations with advanced computing power provides emergency responders with a valuable tool to make well-informed decisions during fire incidents. By leveraging the insights offered by CFD simulations, responders can optimize their strategies, enhancing safety measures and minimizing the consequences of fire incidents.

7.4. Multi-Physics and Multi-Scale Modeling

Multi-physics and multi-scale modeling are essential in advancing Computational Fluid Dynamics (CFD) studies, especially in fire dynamics. The integration of these modeling approaches allows for a comprehensive understanding of complex fire phenomena, enabling researchers to explore interactions across different scales and physical domains.

Kim [102] introduced a Multi-Scale, Multi-Dimensional model for analyzing lithium-ion batteries, resolving electrochemical, electrical, and thermal-coupled physics in battery designs across varied length scales. This approach displays the importance of considering multiple physics phenomena in intricate systems like batteries [102].

In the realm of fire safety, ref. [103] utilized FDS version6.7.0, a Computational Fluid Dynamics (CFD) model of turbulence LES (Large Eddy Simulation) to investigate fire spread from an initial apartment to the overall façade, demonstrating the application of multi-physics modeling in understanding fire propagation behavior [103]. Similarly, Morvan & Dupuy (2004) [104] explored the propagation of wildfires through a Mediterranean shrub using a multiphase formulation, highlighting the complexity of integrating multi-physics considerations in fire spread simulations [104].

Furthermore, the study by [105] focused on the scalability of a multi-physics system for forest fire spread prediction, emphasizing the importance of considering atmosphere-fire interactions in fire modeling. This research underscores the significance of multi-physics modeling in predicting and understanding fire behavior in complex environmental scenarios [105].

In conclusion, the integration of multi-physics and multi-scale modeling techniques in CFD studies for fires provides a comprehensive framework for analyzing and predicting fire dynamics. By considering interactions across different scales and physical domains, researchers can gain valuable insights into fire behavior, aiding in the development of effective fire safety strategies.

7.5. Sustainable and Energy-Efficient Smoke Removal Solutions

In the pursuit of sustainable fire safety solutions, Computational Fluid Dynamics (CFD) studies play a pivotal role in developing efficient smoke removal strategies, particularly in the context of fires involving electric cars. By integrating multi-physics and multi-scale modeling approaches, researchers can gain valuable insights into smoke dispersion dynamics and ventilation systems, leading to the design of sustainable and energy-efficient solutions.

Węgrzyński [105] conducted numerical analyses to optimize smoke removal systems for buildings and road tunnels, focusing on predicting smoke flow patterns in various environments. Their work underscores the importance of utilizing predictive modeling to enhance smoke removal efficiency [106].

One of the smoke removal techniques is High-Efficiency Particulate Air (HEPA) filters. These filters can capture fine particles like smoke and lithium-ion battery aerosols without releasing them back into the environment. They are reusable after proper cleaning and decontamination. The study made by Yu [107] demonstrates the capability of HEPA filters

to trap metal particles released during electrical discharge processes, displaying their versatility in various applications beyond air filtration [107].

The second solution for smoke removal is electrostatic precipitators. They are efficient air pollution control devices that use an electrical charge to attract and trap smoke particles, providing an energy-efficient alternative to traditional filtration systems. Research by Kawada and Shimizu [108] supports the effectiveness of electrostatic precipitators in handling large gas flow rates due to their low-pressure drop compared to other types of precipitators, contributing to their energy efficiency, and making them a preferred choice for industrial applications requiring high-volume air treatment [108].

Solar/wind-powered ventilation systems can provide a sustainable solution for removing smoke from parking areas while utilizing renewable energy sources. Chen [109] proposed an integrated energy system for parks that considers wind and solar energy consumption, emphasizing the importance of optimizing energy utilization in such environments. This approach aligns with the concept of utilizing renewable energy sources like solar power to enhance the sustainability of ventilation systems in parking areas.

In conclusion, sustainable and energy-efficient smoke removal solutions in CFD studies for fires are essential for ensuring environmental safety and minimizing the impact of fire incidents. By leveraging advanced modeling techniques and innovative technologies, researchers can develop effective strategies to control smoke dispersion, improve indoor air quality, and enhance overall fire safety measures in various settings.

7.6. Collaborative and Interdisciplinary Research

Collaborative and interdisciplinary research in the field of Computational Fluid Dynamics (CFD) for fire-related studies involves a combination of various scientific disciplines to enhance the understanding of fire dynamics and safety measures. By integrating CFD simulations with experimental data, researchers can validate the accuracy of their models [110]. This approach allows for a detailed analysis of fire behavior in different scenarios, from small-scale setups to large-scale fires [111]. The use of CFD models as research tools provides insights into fire dynamics, combustion, and smoke behavior [112].

Interdisciplinary collaboration is crucial in fire research, as demonstrated by workshops like the one organized by the IAFSS Working Group on Measurement and Computation of Fire Phenomena (MaCFP) [108]. Such initiatives aim to improve integration and coordination in fire research, emphasizing the importance of model validation for accurate results [113].

Moreover, collaborative efforts between research institutions, such as the VTT Technical Research Centre of Finland and the National Institute of Standards and Technology (NIST), have led to the development of open-source fire-driven fluid flow models [114].

Overall, the synthesis of CFD modeling, interdisciplinary collaboration, and experimental validation plays a pivotal role in advancing fire safety research. By combining expertise from various fields such as engineering, physics, materials science, and chemistry, researchers can develop comprehensive models to analyze fire behavior, improve safety measures, and mitigate risks effectively.

Progress in fire safety science significantly relies on the utilization of Computational Fluid Dynamics (CFD) to replicate a diverse array of scenarios encompassing intricate geometries, multiple length/time scales, and multi-physics phenomena such as turbulence, combustion, heat transfer, soot generation, solid pyrolysis, flame spread, and liquid evaporation. These scenarios are challenging to investigate using traditional analytical solutions and zone models [115]. CFD simulations play a crucial role in fire safety engineering by enabling the examination of fire dynamics in various settings, including compartment fires, tunnel fires, and even scenarios involving leaked gases [116,117]. The use of CFD models has become increasingly prevalent in analyzing fire sprinkler systems, where the behavior of sprinkler sprays is simulated to enhance fire safety measures [118].

Moreover, the integration of CFD simulations with structural response analyses is essential for comprehensive fire safety design, particularly in scenarios like jet fires where

the interaction between fire loads and varying geometry and material properties must be considered [119]. Additionally, CFD models are instrumental in evaluating fire protection systems like water spray systems in confined compartments, aiding in enhancing prediction capabilities and overall system performance [120]. Furthermore, CFD methodologies are pivotal in assessing flame propagation characteristics over surfaces like cladding walls, contributing to a performance-based engineering approach for fire hazard evaluation [121].

In conclusion, the application of CFD in fire safety science has revolutionized the field by allowing for detailed simulations of complex fire dynamics, aiding in the design of effective fire protection systems, and facilitating a deeper understanding of fire behavior in diverse environments.

Future research should prioritize modeling fire size and spread in underground parking facilities, as these factors directly influence the effectiveness of smoke management and evacuation protocols. Understanding fire dynamics in various fire scenarios will lead to the development of better firefighting strategies and improved fire safety designs.

8. Statistics

To provide a better understanding of the methodologies and tools used in fire simulation studies with a special focus on confined spaces and novel problems such as electric car fires, an evaluation of 108 studies was performed, focusing on 46 articles used for statistical data aiming to analyze in an integrated way, the crucial aspects of fire safety and smoke evacuation in underground parking. This integrated selection ensures that the statistical analysis covers the technical details of CFD modeling and also considers the real-world impact and evolving safety standards, creating a base foundation for understanding and improving fire safety in underground parking.

The chart from Figure 10 visualizes the frequency of different software tools utilized in the fire simulation studies reviewed. FDS is the most used software, indicating a preference for its capabilities in modeling complex fire scenarios due to its robustness in simulating fire-driven fluid flow and smoke movement, which is crucial in underground parking scenarios. Software tools like ANSYS (including CFX and Fluent), and Autodesk CFD are moderately used, highlighting their application in various fire simulation contexts, but with specific limitations or specializations compared to FDS. The lower frequency of software like OPEN FOAM, MFIRE, PHOENICS, and SIM SCALE suggests these tools might be used for niche applications or less common fire simulation scenarios. Overall, the chart illustrates the diversity in software tools employed, reflecting the varied approaches and methodologies in fire safety research.

The statistical data from Table 2 was performed taking into consideration the object of study for identification of the environment type also identified the fire regime considered in simulations used was also identified together with the numerical method and the type of mesh (structured mesh or unstructured mesh).

Table 2. Statistical summary of the methodologies employed in fire simulations.

Methodologies	Steady-State		Transient		Total
	Structured Mesh	Unstructured Mesh	Structured Mesh	Unstructured Mesh	
LES			26		26
RANS	2	2	3	3	10
URANS			4	4	8
VLES			1		1
HARDY CROSS			1		1
TOTAL	2	2	35	7	46

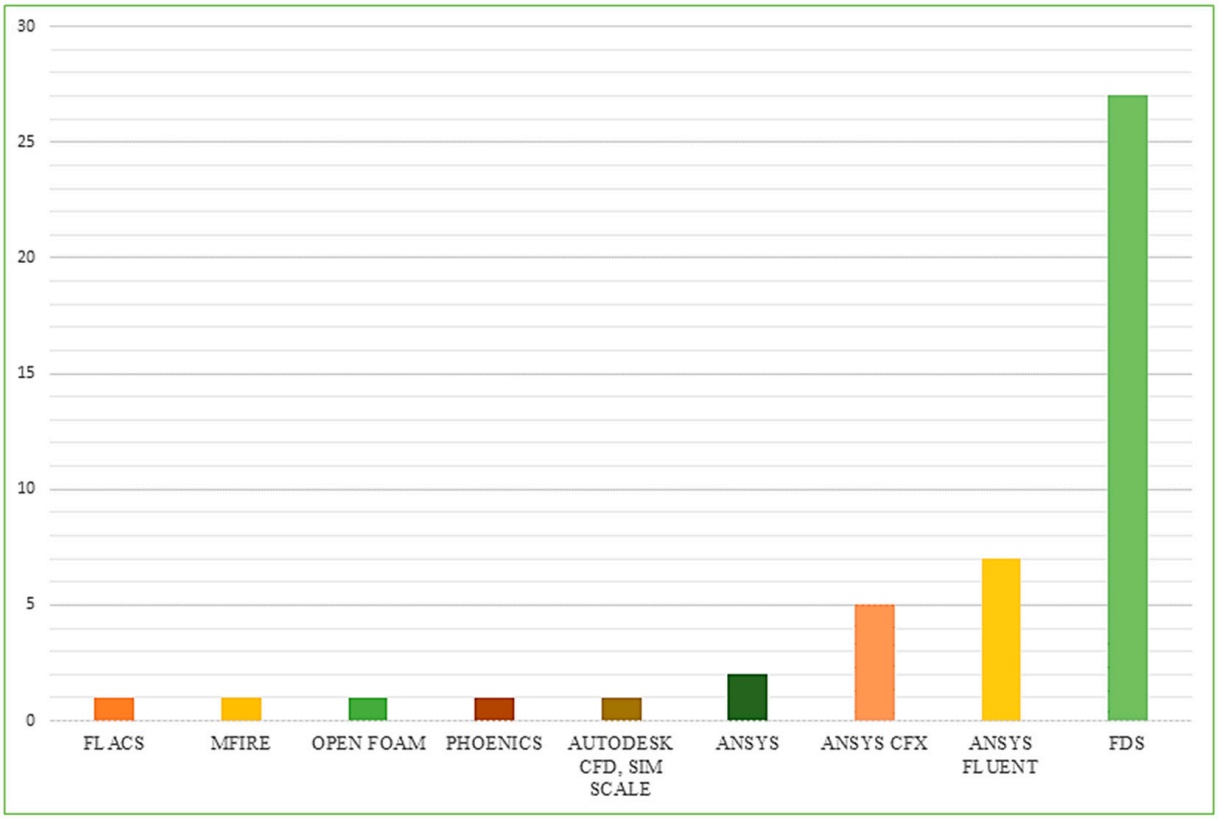


Figure 10. Statistical chart with software types used in articles.

The concept of fire regimes is an essential tool in understanding and managing fire-prone ecosystems. However, the theoretical fire regimes often contrast with the complexities and unpredictability of real-world scenarios. In a steady-state regime fire simulation, the fires occur at regular, predictable intervals with consistent intensity and effects in contrast to transient regime fire simulations, where the fires exhibit significant variability over time, influenced by changing conditions. In summary, the transient fire regime more accurately reflects the reality of fires in buildings due to its acknowledgment of variability, unpredictability, and the influence of multiple dynamic factors. Effective fire safety strategies should therefore be designed to accommodate the complexities and uncertainties inherent in real-world fire scenarios. From Table 3, the most studied fire regimes are those of the transient regime type, which indicates that the data obtained from simulations of this type are closer to the behavior of fire and deserve to be considered useful in future research.

In fire studies, the type of mesh used plays a very important role in determining the accuracy and reliability of simulations and experiments. A fine mesh provides higher resolution and more accurate results, capturing detailed features of the fire behavior such as flame structure, heat transfer, and fluid dynamics, but it requires significantly more computational resources, while the course mesh is less computationally demanding but can miss finer details, potentially leading to less accurate results, but is useful for larger scale simulations where capturing detailed behavior is not as critical. A structured mesh consists of a regular grid pattern, which is easier to implement and requires less computational effort, but it may not be as flexible in representing complex geometries while unstructured mesh is more flexible in handling complex geometries, but it can be computationally intensive due to the need for more complex data structures and algorithms.

Table 3. Key aspects discussed in the document, focusing on the findings, areas for improvement, recommendations for future action, and the potential risks and limitations associated with achieving the objectives set out in the study.

Category	Details
Findings	- CFD simulations are critical in enhancing fire safety by improving smoke exhaust efficiency in underground parking environments. - EV fires pose unique challenges due to their high-voltage battery systems, which can lead to thermal runaway events. - Jet fan systems are effective in smoke removal, especially when combined with conventional ventilation systems.
Improvements Needed	- Enhance the precision of input parameters for CFD simulations to increase reliability. - Develop more robust empirical validation methods to verify CFD models.
Recommendations	- Integrate real-time data and smart technologies to optimize fire safety strategies. - Conduct further research on the fire characteristics of electric vehicles to develop more effective smoke management strategies. - Implement combined jet fan and conventional ventilation systems in underground parking structures to enhance smoke removal efficiency. - Integrate real-time data and smart technologies to optimize fire safety strategies.
Recommendations Risks and Limitations	- The effectiveness of CFD simulations is limited by the quality of input data and computational resources. - The unknown risk profile and combustion behavior of lithium-ion batteries in EVs pose challenges to accurate fire simulation. - Potential for high computational costs and time demands in conducting detailed CFD simulations. - Risk of inadequate smoke management if CFD models are not accurately validated against empirical data - The effectiveness of CFD simulations is limited by the quality of input data and computational resources. - The unknown risk profile and combustion behavior of lithium-ion batteries in EVs pose challenges to accurate fire simulation. - Potential for high computational costs and time demands in conducting detailed CFD simulations. - Risk of inadequate smoke management if CFD models are not accurately validated against empirical data

According to Table 2, LES is the most frequently used method, primarily in structured mesh configurations, with a consistent application across both steady-state and transient simulations; on the other hand, RANS and URANS methods are used less frequently, with a balanced distribution between structured and unstructured meshes in transient simulations.

In our studies, the most used type of mesh was the structured type with 80%, but choosing the appropriate mesh type depends on balancing the need for detail with the available resources and the specific objectives of the study.

In accordance with Figure 11, the largest portion of studies focused on car parks, accounting for 44% of the total. This high percentage reflects the critical need to understand fire dynamics in these spaces, especially considering the unique challenges posed by confined spaces and the presence of vehicles, especially EVs, which can serve as fuel sources and create complex smoke and heat distribution patterns. Tunnels present significant fire safety challenges due to their confined nature, limited evacuation routes, and potential for rapid smoke spread, making them a crucial area of study, and are the second most common focus, comprising 28% of the studies. Studies on buildings account for 26% of the total. This category includes a variety of structures, ranging from residential to commercial buildings, each with different fire safety concerns, such as the need for effective evacuation routes, smoke management, and the protection of occupants.

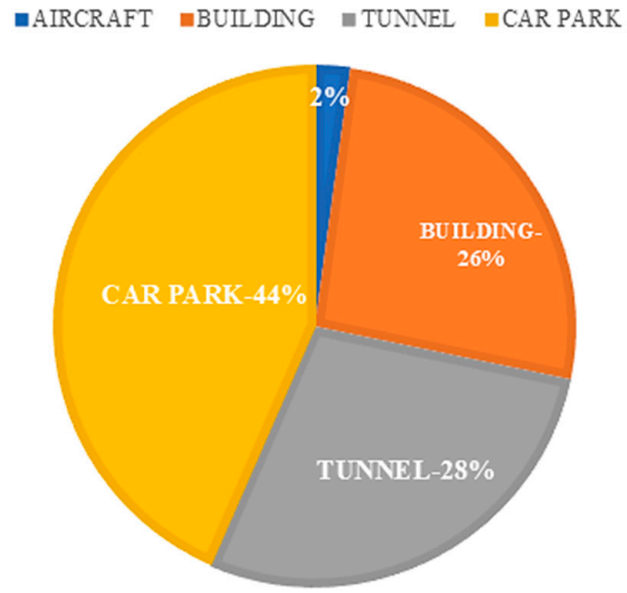


Figure 11. Statistical chart with the object of study used in the articles.

9. Conclusions

The development of fire safety protocols in parking lots has grown more intricate and crucial with the rise in electric vehicle (EV) usage and the unique challenges they bring. Fire incidents happening in such environments have shown the importance of having advanced smoke exhaust systems and a deeper understanding of fire dynamics specific to both conventional and electric vehicles. Adopting Computational Fluid Dynamics (CFD) simulations has become an important development in this field, providing a good method for efficiently analyzing and optimizing smoke exhaust and improving overall fire safety strategies.

Key areas investigated include the hazards of EV fires and thermal runaway mode events or subsequent toxic gases produced during the fire phase (research on these topics shape ventilation design and emergency strategies). The results underscore the importance of conducting empirical studies and developing guidelines for testing to further improve our understanding and prevention strategies about vehicle fires, especially those involving EVs. The report also indicates a shift towards incorporating smart technologies in fire safety practices powered by predictive analytics, using data gathered from previous disasters for better prevention, mitigation, and evacuation policies.

Overall, the next steps in research would be to take a more well-rounded approach that unifies CFD simulations with developments made within material science and environmental studies as real-time monitoring systems are implemented. It aims to develop new fire safety solutions that are more robust and flexible to keep up with changing technologies used for vehicle construction and urban infrastructure. Solutions can be found through interdisciplinary collaboration to address the knowledge and regular gaps, thus maintaining a high level of fire safety in underground parking facilities. To sum up, there is a critical need for efficient fire ventilation systems and procedures to cope with the high heat release rate values of modern vehicle fires. As with other compact, high-rise structures like automated multi-story parking garages that are within the land-use umbrella of “enclosed space”, if such fires are not rendered extremely nuisance-level by managing them as serious life threats from inception through post-operation investigation and site analysis phases—especially where new/on-going fire conditions concerns—not just misleading/exaggeration then lives almost certainly will be lost. In an ever-changing automotive technology landscape, it follows that the strategies and technologies used to keep us safe from vehicle fires in urban areas will need to change as well.

The accuracy of CFD simulation results depends on the quality of the input data, including the fidelity of numerical parameters and the appropriate selections or omission of turbulence models. These factors together determine the precision of the output. It is even more complicated when it comes to simulating electric vehicle fires because the risk profile and fire characteristics that are not well known with lithium-ion batteries make it difficult to simulate them accurately or predict their behavior in a fire.

Thus, the future of CFD technology is also in combining interdisciplinary research and working with experts from engineering, computer science, environment, and emergency management. This sort of collaboration can lead to new, high-tech developments in smoke removal systems that are not only very advanced on the technological front but also practical and user-friendly—smoothly fitting in a mosaic of fire safety and building management solutions.

A unique aspect of this study was the identification of requirements that needed to be fulfilled to ensure the credibility and dependability of CFD simulations. These requirements included the importance of having high-quality input data accurately modeling phenomena and relying on the expertise of those conducting the simulations. Another innovative element highlighted in this analysis was the suggestion to incorporate real-time data and smart technologies into CFD simulations. This integration aimed to enhance the precision and trustworthiness of smoke management strategies by offering adaptable solutions to fire incidents occurring in parking structures. Furthermore, this study offered a viewpoint on the conditions for reliable CFD simulations, particularly considering contemporary challenges like electric vehicle fires in underground parking facilities.

The future of research in CFD for fire scenarios presents significant opportunities for improvement, especially in integrating emerging technologies such as machine learning (ML) and artificial intelligence (AI). By leveraging ML techniques, future models can be optimized to enhance accuracy and reduce computation time, which is particularly critical in real-time fire safety applications. The development of hybrid systems combining CFD simulations with AI-driven predictive analytics can significantly improve the precision of fire simulations, especially in complex environments like underground parking facilities. These advancements can lead to more responsive systems capable of adjusting to real-time conditions during a fire, optimizing smoke extraction, and improving overall safety strategies. Additionally, ongoing interdisciplinary collaboration between engineers, computer scientists, and fire safety experts will be very important in developing more robust, scalable solutions that can keep pace with advancements in vehicle technology, including the rise of electric vehicles (EVs). These advancements are likely to enhance the precision of CFD models, making them more adaptable to new fire challenges such as EV-related fires, where the unique risks posed by lithium-ion batteries are still not fully understood.

Funding: This work was supported by a grant of the Ministry of Research, Innovation, and Digitization, CCCDI—UEFISCDI, project number ENGINE PN-IV-P8-8.1-PRE-HE-ORG-2023-0161.

Conflicts of Interest: The authors declare no conflicts of interest.

References

1. Shen, R.; Jiao, Z.; Parker, T.; Sun, Y.; Wang, Q. Recent application of Computational Fluid Dynamics (CFD) in process safety and loss prevention: A review. *J. Loss Prev. Process Ind.* **2020**, *67*, 104252. [CrossRef]
2. Lazarev, Y.; Bashkarev, A.; Makovetskaya-Abramova, O.; Amirseyidov, S. Study of technical reasons for fire-burning vehicles. *E3S Web Conf.* **2023**, *389*, 05053. [CrossRef]
3. Zhu, H.; Gao, Y.; Guo, H. Experimental investigation of burning behavior of a running vehicle. *Case Stud. Therm. Eng.* **2020**, *22*, 100795. [CrossRef]
4. Lönnermark, A.; Blomqvist, P. Emissions from an automobile fire. *Chemosphere* **2006**, *62*, 1043–1056. [CrossRef]
5. Sun, P.; Bisschop, R.; Niu, H.; Huang, X. A Review of Battery Fires in Electric Vehicles. *Fire Technol.* **2020**, *56*, 4. [CrossRef]
6. Another-Major-Car-Park-Fire-In-Paris. Available online: https://www.eurosprinkler.org/another-major-car-park-fire-in-paris (accessed on 11 January 2024).

7. Massive-Parking-Fire-Luton-Airport-Likely-Was-Started-Diesel-Car-Not-Ev. Available online: https://www.ctif.org/news/massive-parking-fire-luton-airport-likely-was-started-diesel-car-not-ev (accessed on 11 January 2024).
8. Structural-Integrity-Of-Garage-Now-In-Question-Following-Fire-At-Stamford-Conn-Hotel. Available online: https://www.cbsnews.com/newyork/news/structural-integrity-of-garage-now-in-question-following-fire-at-stamford-conn-hotel (accessed on 11 January 2024).
9. Stamford-Marriott-Hotel-Cars-Fire-Garage-17894815.php. Available online: https://www.stamfordadvocate.com/news/article/stamford-marriott-hotel-cars-fire-garage-17894815.php (accessed on 11 January 2024).
10. Grafic vanzari EV. Available online: https://www.iea.org/energy-system/transport/electric-vehicles (accessed on 11 January 2024).
11. Winkler, S.L.; Anderson, J.E.; Garza, L.; Ruona, W.C.; Vogt, R.; Wallington, T.J. Vehicle criteria pollutant (PM, NOx, CO, HCs) emissions: How low should we go? *npj Clim. Atmos. Sci.* **2018**, *1*, 26. [CrossRef]
12. Merci, B.; Shipp, M. Smoke and heat control for fires in large car parks: Lessons learnt from research? *Fire Saf. J.* **2013**, *57*, 3–10. [CrossRef]
13. Burlacu, D.; Anghel, I.; Popa, C.; Căşaru, I. Fire Safety Evaluation of an Underground Car Park Using Numerical Simulation. *Math. Model. Civ. Eng.* **2018**, *14*, 1–12. [CrossRef]
14. Fire EV in Belgia. Available online: https://autolatest.ro/stiri/pompierii-din-belgia-vor-sa-interzica-accesul-masinilor-electrice-in-parcarile-subterane/ (accessed on 11 January 2024).
15. EV Banned in Parkings. Available online: https://www.infranken.de/lk/kulmbach/tiefgarage-e-autos-muessen-draussen-bleiben-art-5164055 (accessed on 11 January 2024).
16. Tivey, J.; Davies, H.C.; Levine, J.G.; Zietsman, J.; Bartington, S.; Ibarra-Espinosa, S.; Ropkins, K. Meta-Analysis as Early Evidence on the Particulate Emissions Impact of EURO VI on Battery Electric Bus Fleet Transitions. *Sustainability* **2023**, *15*, 1522. [CrossRef]
17. Lulea, M.D.; Iordache, V.; Năstase, I. Numerical Model Development of the Air Temperature Variation in a Room Set on Fire for Different Ventilation Scenarios. *Appl. Sci.* **2021**, *11*, 11698. [CrossRef]
18. Nakamura, Y.; Hajjawi, M. Underground Car Park Smoke Management System Design Validation Using CFD Simulation: Car Fire Products Yields Rates system. In Proceedings of the 8th World Congress on Mechanical, Chemical, and Material Engineering, Prague, Czech Republic, 31 July–2 August 2022. [CrossRef]
19. Dorsz, A.; Lewandowski, M. Analysis of Fire Hazards Associated with the Operation of Electric Vehicles in Enclosed Structures. *Energies* **2021**, *15*, 11. [CrossRef]
20. Jin, X.; Zhang, L.; Gong, J.; Zhu, C.; Wan, H.; Li, X. Smoke Prevention and Exhaust System for Buildings Based on Performance-Oriented Design. *IJDNE* **2020**, *15*, 805–812. [CrossRef]
21. Kmecová, M.; Krajčík, M.; Straková, Z. Designing Jet Fan Ventilation for an Underground Car Park by CFD Simulations. *Period. Polytech. Mech. Eng.* **2018**, *63*, 39–43. [CrossRef]
22. Liu, Q.; Xiao, J.; Cai, B.; Guo, X.; Wang, H.; Chen, J.; Zhang, M.; Qiu, H.; Zheng, C.; Zhou, Y. Numerical Simulation on the Effect of Fire Shutter Descending Height on Smoke Extraction Efficiency in a Large Atrium. *Fire* **2022**, *5*, 101. [CrossRef]
23. Ahn, S.-J.; Kwon, H.-M.; Kim, G.-H.; Yang, J.-H. Study of Securing Required Ventilation Rates and Improving Mechanical Ventilation Systems for Underground Parking Lots. *J. Asian Archit. Build. Eng.* **2016**, *15*, 659–665. [CrossRef]
24. Tilley, N.; Deckers, X.; Merci, B. CFD study of relation between ventilation velocity and smoke backlayering distance in large closed car parks. *Fire Saf. J.* **2012**, *48*, 11–20. [CrossRef]
25. Fang, Z.-M.; Song, W.-G.; Zhang, J.; Wu, H. A Multi-Grid Model for Evacuation Coupling with the Effects of Fire Products. *Fire Technol.* **2012**, *48*, 91–104. [CrossRef]
26. Deckers, X.; Haga, S.; Tilley, N.; Merci, B. Smoke control in case of fire in a large car park: CFD simulations of full-scale configurations. *Fire Saf. J.* **2013**, *57*, 22–34. [CrossRef]
27. Nazari, A.; Jafari, M.; Rezaei, N.; Taghizadeh-Hesary, F.; Taghizadeh-Hesary, F. Jet fans in the underground car parking areas and virus transmission. *Phys. Fluids* **2021**, *33*, 1. [CrossRef]
28. Deckers, X.; Haga, S.; Sette, B.; Merci, B. Smoke control in case of fire in a large car park: Full-scale experiments. *Fire Saf. J.* **2013**, *57*, 11–21. [CrossRef]
29. Węgrzyński, W.; Krajewski, G.; Kimbar, G. Smart Smoke Control as an Efficient Solution for Smoke Ventilation in Converted Cellars of Historic Buildings. *Fire Technol.* **2021**, *57*, 3101–3123. [CrossRef]
30. Chen, F.; Leong, J.C. Smoke flow phenomena and turbulence characteristics of tunnel fires. *Appl. Math. Model.* **2011**, *35*, 4554–4566. [CrossRef]
31. Chen, Z.; Wen, J.; Xu, B.; Dembele, S. Extension of the eddy dissipation concept and smoke point soot model to the LES frame for fire simulations. *Fire Saf. J.* **2014**, *64*, 12–26. [CrossRef]
32. Wang, Q.; Tang, F.; Li, L.; Zhang, X.; Fan, C. Large Eddy Simulation on the Effect of Smoke Exhaust Openings Arrangement on the Smoke Spread in Tunnel Fires. *Procedia Eng.* **2016**, *135*, 309–315. [CrossRef]
33. Wang, Q.; Zhou, T.; Liu, Q.; He, P.; Tao, C.; Shi, Q. Numerical study of critical re-entrainment velocity of fire smoke within the street canyons with different building height ratios. *Env. Sci. Pollut. Res.* **2019**, *26*, 23319–23327. [CrossRef]
34. Wen, H.; Liu, Y.; Guo, J.; Zhang, Z.; Liu, M.; Cai, G. Study on Numerical Simulation of Fire Danger Area Division in Mine Roadway. *Math. Probl. Eng.* **2021**, *2021*, 6646632. [CrossRef]

35. Edgar, R.A.; Sharples, J.J.; Sidhu, H.S. Revisiting the King's Cross Underground disaster with implications for modelling wildfire eruption. In Proceedings of the MODSIM2015, 21st International Congress on Modelling and Simulation, Modelling and Simulation Society of Australia and New Zealand, Gold Coast, Australia, 29 November–4 December 2015. [CrossRef]
36. Rahif, R.; Attia, S. CFD Assessment of Car Park Ventilation System in Case of Fire Event. *Appl. Sci.* **2023**, *13*, 10190. [CrossRef]
37. Cheng, L.H.; Ueng, T.H.; Liu, C.W. Simulation of ventilation and fire in the underground facilities. *Fire Saf. J.* **2001**, *36*, 597–619. [CrossRef]
38. Liang, Y.; Yang, Y.; Guo, S.; Tian, F.; Wang, S. Combustion mechanism and control approaches of underground coal fires: A review. *Int. J. Coal Sci. Technol.* **2023**, *10*, 24. [CrossRef]
39. Yao, H.-W.; Lv, K.-F.; Li, Y.-X.; Zhang, J.-G.; Lv, Z.-B.; Wang, D.; Zhan, Z.-Y.; Wei, X.-G.; Song, H.-T.; Qin, H.-J. Numerical Simulation of Fire in Underground Commercial Street. *Comput. Intell. Neurosci.* **2022**, *2022*, 4699471. [CrossRef]
40. Annerel, E.; Taerwe, L.; Merci, B.; Jansen, D.; Bamonte, P.; Felicetti, R. Thermo-mechanical analysis of an underground car park structure exposed to fire. *Fire Saf. J.* **2013**, *57*, 96–106. [CrossRef]
41. Khalil, E.; Fouad, M.; Mohamed, S. Effect of Fire Locations on the Performance of Impulse Ventilation System in an Underground Car Park. In *51st AIAA Aerospace Sciences Meeting Including the New Horizons Forum and Aerospace Exposition*; American Institute of Aeronautics and Astronautics: Grapevine, TX, USA, 2013. [CrossRef]
42. Quaife, B.; Speer, K. A Simple Model for Wildland Fire Vortex–Sink Interactions. *Atmosphere* **2021**, *12*, 1014. [CrossRef]
43. Chi, J.; Wu, S.; Shu, C. Using Fire Dynamics Simulator to Reconstruct a Hydroelectric Power Plant Fire Accident. *J. Forensic Sci.* **2011**, *56*, 1639–1644. [CrossRef] [PubMed]
44. Xu, M.; Bu, P.; Xin, L.; Li, C.; Han, L.; An, M.; Feng, M.; Li, K. Research on factors affecting bus fire based on numerical simulation. *E3S Web Conf.* **2021**, *303*, 01042. [CrossRef]
45. Available online: https://evmarketsreports.com/ev-fire-incidents-a-manageable-risk-amidst-ev-surge/ (accessed on 15 June 2024).
46. Truchot, B.; Fouillen, F.; Collet, S. An experimental evaluation of toxic gas emissions from vehicle fires. *Fire Saf. J.* **2018**, *97*, 111–118. [CrossRef]
47. Tramoni, J.; Thauvoye, C.; Hanus, F.; Poutrain, B.; Suzanne, M.; Thiry, A. Temperature assessment of steel members subjected to fire generated by alternative fuel vehicles: Experimental tests. *Fire Mater.* **2021**, *45*, 1096–1105. [CrossRef]
48. Kang, S.; Lee, K.M.; Kwon, M.; Lim, O.K.; Choi, J.Y. A Quantitative Analysis of the Fire Hazard Generated from Hydrogen Fuel Cell Electric Vehicles. *Int. J. Fire Sci. Eng.* **2022**, *36*, 26–39. [CrossRef]
49. Maragkos, G.; Beji, T. Review of Convective Heat Transfer Modelling in CFD Simulations of Fire-Driven Flows. *Appl. Sci.* **2021**, *11*, 5240. [CrossRef]
50. Aminfar, A.; Cobian-Iñiguez, J.; Ghasemian, M.; Espitia, N.R.; Weise, D.R.; Princevac, M. Using Background-Oriented Schlieren to Visualize Convection in a Propagating Wildland Fire. *Combust. Sci. Technol.* **2020**, *192*, 2259–2279. [CrossRef]
51. Florea, G.D.; Vlasin, N.I.; Şimon-Marinică, A.B.; Manea, F.; Vass, Z. Computational simulation of heat conduction on different surfaces. *MATEC Web Conf.* **2021**, *342*, 01014. [CrossRef]
52. Xie, Y.; Liu, X.; Zhang, C.; Zhao, J.; Wang, H. Coupled heat transfer model for the combustion and steam characteristics of coal-fired boilers. *Eng. Appl. Comput. Fluid Mech.* **2021**, *15*, 490–502. [CrossRef]
53. La Scala, A.; Loprieno, P.; Foti, D.; La Scala, M. The Mechanical Response of Structural Elements in Enclosed Structures during Electric Vehicle Fires: A Computational Study. *Energies* **2023**, *16*, 7233. [CrossRef]
54. Vega, M.G.; Díaz, K.M.A.; Oro, J.M.F.; Tajadura, R.B.; Morros, C.S. Numerical 3D simulation of a longitudinal ventilation system: Memorial Tunnel case. *Tunn. Undergr. Space Technol.* **2008**, *23*, 539–551. [CrossRef]
55. Se, C.M.K.; Lee, E.W.M.; Lai, A.C.K. Impact of location of jet fan on airflow structure in tunnel fire. *Tunn. Undergr. Space Technol.* **2012**, *27*, 30–40. [CrossRef]
56. Kim, J.-H.; Kim, J.-H.; Kim, K.-Y.; Yoon, J.-Y.; Yang, S.-H.; Choi, Y.-S. High-efficiency design of a tunnel ventilation jet fan through numerical optimization techniques. *J. Mech. Sci. Technol.* **2012**, *26*, 1793–1800. [CrossRef]
57. Immonen, E. CFD optimization of jet fan ventilation in a car park by fractional factorial designs and response surface methodology. *Build. Simul.* **2016**, *9*, 53–61. [CrossRef]
58. Kazemipour, A.; Afshin, H.; Farhanieh, B. Numerical–Analytical Assessment of Fire and Ventilation Interaction in Longitudinally Ventilated Tunnels Using Jet Fans. *Heat Transf. Eng.* **2017**, *38*, 523–537. [CrossRef]
59. Khieu, H.T.; Lee, Y.M.; Kim, J.T.; Ryou, H.S. Numerical Study of the Effects of the Jet Fan Speed, Heat Release Rate and Aspect Ratio on Smoke Movement in Tunnel Fires. *Energies* **2020**, *13*, 1206. [CrossRef]
60. Lu, S.; Wang, Y.H.; Zhang, R.F.; Zhang, H.P. Numerical Study on Impulse Ventilation for Smoke Control in an Underground Car Park. *Procedia Eng.* **2011**, *11*, 369–378. [CrossRef]
61. Senveli, A.; Dizman, T.; Celen, A.; Bilge, D.; Dalkılıç, A.S.; Wongwises, S. CFD Analysis of Smoke and Temperature Control System of an Indoor Parking Lot with Jet Fans. *J. Therm. Eng.* **2015**, *1*, 116. [CrossRef]
62. Borowski, M.; Jaszczur, M.; Karch, M.; Burdzy, T. Control of smoke flow using a jet-fan in an underground car park. *E3S Web Conf.* **2019**, *128*, 10007. [CrossRef]
63. Çakir, M.T.; Ün, Ç. CFD Analysis of Smoke and Temperature Control System of Car Park Area with Jet Fans. *J. Eng. Res. Rep.* **2020**, *13*, 27–40. [CrossRef]

64. Balan, M.C.; Verdeș, M.; Ciocan, V.; Hudișteanu, S.V.; Țurcanu, F.E.; Burlacu, A.; Grosu, A. 21. PERFORMANCE OF VENTILATION AND SMOKE EXHAUST SYSTEMS IN CASE OF FIRE IN UNDERGROUND PARK-ING LOTS: CASE STUDY. In Proceedings of the International Conference Building Services and Energy Efficiency, Frankfurt, Germany, 12–13 March 2020; pp. 241–256. [CrossRef]
65. Sultansu, S.; Onat, A. The Cfd Analysis of Ventilation and Smoke Control System with Jet Fan in a Parking Garage. *Int. J. Adv. Eng. Pure Sci.* **2020**, *32*, 89–95. [CrossRef]
66. Alianto, B.; Astari, N.; Nareshwara, D.; Nugroho, Y.S. Modeling of Smoke Control in Underground Parking-garage Fires. *Int. J. Technol.* **2017**, *8*, 7. [CrossRef]
67. Datciuk, T.; Vasil'ev, V.; Ulyasheva, V. Analysis of the state of the air environment in the underground parking. *E3S Web Conf.* **2023**, *371*, 02008. [CrossRef]
68. Jiang, Y.; Zhang, B.; Huang, T. CFD Study of an Annular-Ducted Fan Lift System for VTOL Aircraft. *Aerospace* **2015**, *2*, 555–580. [CrossRef]
69. Kawano, T.; Fuchiwaki, M. Aerodynamic performances and efficiency of axial flow fan placed in cooling system. *J. Phys. Conf. Ser.* **2022**, *2217*, 012002. [CrossRef]
70. Zelenský, P.; Barták, M.; Zavøel, V.; Zmrhal, V.; Krupa, R. Numerical Analysis of Air Flow in a Modular Fan Unit Using CFD Simulation. *E3S Web Conf.* **2019**, *111*, 01008. [CrossRef]
71. Santoso, M.A.; Bey, Z.; Nugroho, Y.S. CFD Study on the Ventilation System and Shape Configuration of Underground Car Park in Case of Fire. *Appl. Mech. Mater.* **2015**, *758*, 143–151. [CrossRef]
72. Fuh, Y.K.; Huang, W.C. A Numerical Study on Operation Modes of Mechanical Ventilation in Underground Tunnels and Comparisons with Full-Scale Experiments and Existing Fire Codes in Taiwan. *Adv. Mater. Res.* **2011**, *243*, 3355–3359. [CrossRef]
73. Gutiérrez-Montes, C.; Sanmiguel-Rojas, E.; Viedma, A.; Rein, G. Experimental data and numerical modelling of 1.3 and 2.3MW fires in a 20m cubic atrium. *Build. Environ.* **2009**, *44*, 1827–1839. [CrossRef]
74. Horváth, I.; Van Beeck, J.; Merci, B. Full-scale and reduced-scale tests on smoke movement in case of car park fire. *Fire Saf. J.* **2013**, *57*, 35–43. [CrossRef]
75. Lock, A.; Bundy, M.; Johnsson, E.L.; Hamins, A.; Ko, G.H.; Hwang, C.; Fuss, P.; Harris, R. *Experimental Study of the Effects of Fuel Type, Fuel Distribution and Vent Size on Full-Scale Underventilated Compartment Fires in an ISO 9705 Room*; NIST TN 1603; National Institute of Standards and Technology: Gaithersburg, MD, USA, 2008. [CrossRef]
76. Sittisak, P.; Charinpanitkul, T.; Chalermsinsuwan, B. Enhancement of carbon monoxide removal in an underground car park using ventilation system with single and twin jet fans. *Tunn. Undergr. Space Technol.* **2020**, *97*, 103226. [CrossRef]
77. Barbato, L.; Cascetta, F.; Musto, M.; Rotondo, G. Fire safety investigation for road tunnel ventilation systems—An overview. *Tunn. Undergr. Space Technol.* **2014**, *43*, 253–265. [CrossRef]
78. Li, J.S.M.; Chow, W.K. Numerical studies on performance evaluation of tunnel ventilation safety systems. *Tunn. Undergr. Space Technol.* **2003**, *18*, 435–452. [CrossRef]
79. Viegas, J.C. The use of impulse ventilation for smoke control in underground car parks. *Tunn. Undergr. Space Technol.* **2010**, *25*, 42–53. [CrossRef]
80. Liu, K.; Liu, Y.; Lin, D.; Pei, A.; Cui, Y. Materials for lithium-ion battery safety. *Sci. Adv.* **2018**, *4*, eaas9820. [CrossRef]
81. Ren, X.; Li, C.; Ma, X.; Chen, F.; Wang, H.; Sharma, A.; Gaba, G.S.; Masud, M. Design of Multi-Information Fusion Based Intelligent Electrical Fire Detection System for Green Buildings. *Sustainability* **2021**, *13*, 3405. [CrossRef]
82. Andersson, P.; Blomqvist, P.; Lorén, A.; Larsson, F. Using Fourier transform infrared spectroscopy to determine toxic gases in fires with lithium-ion batteries. *Fire Mater.* **2016**, *40*, 999–1015. [CrossRef]
83. Moinuddin, K.; Razzaque, Q.S.; Thomas, A. Numerical Simulation of Coupled Pyrolysis and Combustion Reactions with Directly Measured Fire Properties. *Polymers* **2020**, *12*, 2075. [CrossRef]
84. Khattri, S.K.; Log, T.; Kraaijeveld, A. Tunnel Fire Dynamics as a Function of Longitudinal Ventilation Air Oxygen Content. *Sustainability* **2019**, *11*, 203. [CrossRef]
85. Brzezinska, D.; Bryant, P. Performance-Based Analysis in Evaluation of Safety in Car Parks under Electric Vehicle Fire Conditions. *Energies* **2022**, *15*, 649. [CrossRef]
86. Van Der Heijden, M.G.M.; Loomans, M.G.L.C.; Lemaire, A.D.; Hensen, J.L.M. Fire safety assessment of semi-open car parks based on validated CFD simulations. *Build. Simul.* **2013**, *6*, 385–394. [CrossRef]
87. Löhnert, A.; Monreal, N.; Knaust, C.; Hofmann, A.; Krause, U. CFD modeling approach of smoke toxicity and opacity for flaming and non-flaming combustion processes. *Fire Mater.* **2016**, *40*, 759–772. [CrossRef]
88. Gunjal, P.R.; Ranade, V.V.; Chaudhari, R.V. Computational study of a single-phase flow in packed beds of spheres. *AIChE J.* **2005**, *51*, 365–378. [CrossRef]
89. Sun, J.; Fang, Z.; Beji, T.; Merci, B. Interpretation of flow fields induced by water spray systems in reduced-scale tunnel fire experiments by means of CFD simulations. *Tunn. Undergr. Space Technol.* **2018**, *81*, 94–102. [CrossRef]
90. Zhu, Y.; Xiao, F.; Luo, Z. A CFD simulation study to evaluate the flow and catalytic hydrogenation of dimethyl oxalate in a packed bed, a two-stage fluidized bed, and a circulating fluidized bed. *Asia-Pac. J. Chem. Eng.* **2014**, *9*, 280–292. [CrossRef]
91. Brunton, S.L.; Noack, B.R.; Koumoutsakos, P. Machine Learning for Fluid Mechanics. *Annu. Rev. Fluid Mech.* **2019**, *52*, 477–508. [CrossRef]

92. Pathak, J.; Hunt, B.; Girvan, M.; Lu, Z.; Ott, E. Model-Free Prediction of Large Spatiotemporally Chaotic Systems from Data: A Reservoir Computing Approach. *Phys. Rev. Lett.* **2018**, *120*, 024102. [CrossRef]
93. Pishnamazi, M.; Babanezhad, M.; Nakhjiri, A.T.; Rezakazemi, M.; Marjani, A.; Shirazian, S. ANFIS grid partition framework with difference between two sigmoidal membership functions structure for validation of nanofluid flow. *Sci. Rep.* **2020**, *10*, 15395. [CrossRef]
94. Babanezhad, M.; Behroyan, I.; Nakhjiri, A.T.; Marjani, A.; Shirazian, S. Performance and application analysis of ANFIS artificial intelligence for pressure prediction of nanofluid convective flow in a heated pipe. *Sci. Rep.* **2021**, *11*, 902. [CrossRef] [PubMed]
95. Ermolieva, T.; Ermoliev, Y.; Zagorodny, A.; Bogdanov, V.; Borodina, O.; Havlik, P.; Komendantova, N.; Knopov, P.; Gorbacuk, V.; Zaslavskyi, V. Artificial Intelligence, Machine Learning, and Intelligent Decision Support Systems: Iterative "Learning" SQGbased procedures for Distributed Models' Linkage. *Stuc. Intelekt.* **2022**, *27*, 92–97. [CrossRef]
96. Vinuesa, R.; Brunton, S.L. Enhancing Computational Fluid Dynamics with Machine Learning. *Nat. Comput. Sci.* **2022**, *2*, 358–366. [CrossRef] [PubMed]
97. Maragkos, G.; Merci, B. Large Eddy Simulations of CH4 Fire Plumes. *Flow. Turbul. Combust.* **2017**, *99*, 239–278. [CrossRef]
98. Fernánez-Alaiz, F.; Castañón, A.M.; Gómez-Fernández, F.; Bernardo-Sánchez, A.; Bascompta, M. Analysis of the Fire Propagation in a Sublevel Coal Mine. *Energies* **2020**, *13*, 3754. [CrossRef]
99. Alos-Moya, J.; Paya-Zaforteza, I.; Garlock, M.E.M.; Loma-Ossorio, E.; Schiffner, D.; Hospitaler, A. Analysis of a bridge failure due to fire using computational fluid dynamics and finite element models. *Eng. Struct.* **2014**, *68*, 96–110. [CrossRef]
100. Janardhan, R.K.; Shakil, S.; Hassinen, M.; Lu, W.; Puttonen, J.; Hostikka, S. Impact of Firefighting Sprays on the Fire Performance of Structural Steel Members. *Fire Technol.* **2022**, *58*, 2405–2440. [CrossRef]
101. Caliendo, C.; Genovese, G.; Russo, I. Risk Analysis of Road Tunnels: A Computational Fluid Dynamic Model for Assessing the Effects of Natural Ventilation. *Appl. Sci.* **2020**, *11*, 32. [CrossRef]
102. Kim, G.-H.; Smith, K.; Lee, K.-J.; Santhanagopalan, S.; Pesaran, A. Multi-Domain Modeling of Lithium-Ion Batteries Encompassing Multi-Physics in Varied Length Scales. *J. Electrochem. Soc.* **2011**, *158*, A955. [CrossRef]
103. Guillaume, E.; Dréan, V.; Girardin, B.; Koohkan, M.; Fateh, T. Reconstruction of Grenfell Tower fire. Part 2: A numerical investigation of the fire propagation and behaviour from the initial apartment to the façade. *Fire Mater.* **2020**, *44*, 15–34. [CrossRef]
104. Morvan, D.; Dupuy, J.L. Modeling the propagation of a wildfire through a Mediterranean shrub using a multiphase formulation. *Combust. Flame* **2004**, *138*, 199–210. [CrossRef]
105. Farguell, A.; Cortés, A.; Margalef, T.; Miró, J.R.; Mercader, J. Scalability of a multi-physics system for forest fire spread prediction in multi-core platforms. *J. Supercomput.* **2019**, *75*, 1163–1174. [CrossRef]
106. Węgrzyński, W.; Krajewski, G.; Kimbar, G.; Lipecki, T. Fire smoke dispersion inside and outside of a warehouse building in moderate and strong wind conditions. *Fire Saf. J.* **2023**, *136*, 103760. [CrossRef]
107. Yu, B.; Blaesi, A.H.; Casey, N.; Raykhtsaum, G.; Zazzeron, L.; Jones, R.; Morrese, A.; Dobrynin, D.; Malhotra, R.; Bloch, D.B.; et al. Detection and removal of impurities in nitric oxide generated from air by pulsed electrical discharge. *Nitric Oxide* **2016**, *60*, 16–23. [CrossRef]
108. Kawada, Y.; Shimizu, H. Reduction of Suspended Particles in Closed Space with Electrostatic Precipitator. *Elect. Comm. Jpn.* **2017**, *100*, 32–40. [CrossRef]
109. Chen, L.; Wang, Y.; Hu, R.; Xu, G.; Ma, S. Optimal scheduling method of integrated energy system in parks considering wind and solar energy consumption. *J. Phys. Conf. Ser.* **2022**, *2301*, 012012. [CrossRef]
110. Tilley, N.; Rauwoens, P.; Merci, B. Verification of the accuracy of CFD simulations in small-scale tunnel and atrium fire configurations. *Fire Saf. J.* **2011**, *46*, 186–193. [CrossRef]
111. Rengel, B.; Mata, C.; Pastor, E.; Casal, J.; Planas, E. A priori validation of CFD modelling of hydrocarbon pool fires. *J. Loss Prev. Process Ind.* **2018**, *56*, 18–31. [CrossRef]
112. Mueller, E.V.; Skowronski, N.S.; Clark, K.L.; Gallagher, M.R.; Mell, W.E.; Simeoni, A.; Hadden, R.M. Detailed physical modeling of wildland fire dynamics at field scale—An experimentally informed evaluation. *Fire Saf. J.* **2021**, *120*, 103051. [CrossRef]
113. Brown, A.; Bruns, M.; Gollner, M.; Hewson, J.; Maragkos, G.; Marshall, A.; McDermott, R.; Merci, B.; Rogaume, T.; Stoliarov, S.; et al. Proceedings of the first workshop organized by the IAFSS Working Group on Measurement and Computation of Fire Phenomena (MaCFP). *Fire Saf. J.* **2018**, *101*, 1–17. [CrossRef]
114. Caliendo, C.; Genovese, G.; Russo, I. A Simultaneous Analysis of the User Safety and Resilience of a Twin-Tube Road Tunnel. *Appl. Sci.* **2022**, *12*, 3357. [CrossRef]
115. Rein, G.; Torero, J.L.; Jahn, W.; Stern-Gottfried, J.; Ryder, N.L.; Desanghere, S.; Lázaro, M.; Mowrer, F.; Coles, A.; Joyeux, D.; et al. Round-robin study of a priori modelling predictions of the Dalmarnock Fire Test One. *Fire Saf. J.* **2009**, *44*, 590–602. [CrossRef]
116. Kim, E.; Woycheese, J.P.; Dembsey, N.A. Fire Dynamics Simulator (Version 4.0) Simulation for Tunnel Fire Scenarios with Forced, Transient, Longitudinal Ventilation Flows. *Fire Technol.* **2008**, *44*, 137–166. [CrossRef]
117. Ng, Y.W.; Huo, Y.; Chow, W.; Chow, C.L.; Cheng, F.M. Numerical simulations on explosion of leaked liquefied petroleum gas in a garage. *Build. Simul.* **2017**, *10*, 755–768. [CrossRef]
118. Myers, T.M.; Marshall, A.W. A description of the initial fire sprinkler spray. *Fire Saf. J.* **2016**, *84*, 1–7. [CrossRef]
119. Kim, S.J.; Lee, J.; Seo, J.K.; Kim, B.J.; Ha, Y.C.; Paik, J.K.; Lee, K.S.; Park, B.; Ki, M.S.; Sohn, J.M. Nonlinear structural response in jet fire in association with the interaction between fire loads and time-variant geometry and material properties. *Ocean. Eng.* **2017**, *144*, 118–134. [CrossRef]

120. Vaux, S.; Prétrel, H.; Audouin, L. Experimental and numerical study of water spray system for a fire event in a confined and mechanically ventilated compartment. *Fire Mater.* **2019**, *43*, 579–590. [CrossRef]
121. Chen, X.; Wang, F. Performance-based engineering approach to accurately determine flame propagation characteristics over exterior video cladding wall. *Fire Mater.* **2021**, *45*, 690–696. [CrossRef]

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.