OPTIMISING VENTILATION FOR THERMAL COMFORT AND INFECTION CONTROL IN VEHICLES: A REVIEW ON CFD APPLICATIONS AND TECHNIQUES

Kai Ying Tan[a], Sien Jie Wong[a], Huiyi Tan[b], Hong Yee Kek[a], Mohd Hafiz Dzafan Othman[c], Syie Luing Wong[d], Mohamad Nur Hidayat Mat[e], Yi Ka Fong[a] and Keng Yinn Wong[a]

[a]Faculty of Mechanical Engineering, Universiti Teknologi Malaysia, 81310 UTM Johor Bahru, Johor, Malaysia
[b]Faculty of Chemical and Energy Engineering, Universiti Teknologi Malaysia, 81310 UTM Johor Bahru, Johor, Malaysia
[c]Advanced Membrane Technology Research Centre (AMTEC), Universiti Teknologi Malaysia, 81310 UTM Johor Bahru, Johor, Malaysia
[d]Department of Chemical Engineering & Chemistry, Eindhoven University of Technology, 5600 MB, the Netherlands
[e]Faculty of Mechanical and Automotive Engineering Technology, Universiti Malaysia Pahang Al-Sultan Abdullah, 26600 Pekan, Pahang, Malaysia

[*]Corresponding email: kengyinnwong@utm.my

Article history
Received
13th May 2025
Revised
16th June 2025
Accepted
14th August 2025
Published
1st December 2025

ABSTRACT

Effective ventilation in vehicles is crucial for ensuring thermal comfort and controlling airborne infections. This review focuses on the application of Computational Fluid Dynamics (CFD) in optimising vehicle ventilation systems. It examines various CFD methodologies, including turbulence models, meshing techniques, and solver algorithms, as well as their impact on ventilation performance. The review highlights that RNG k-ε and SST k-ω are among the most commonly used turbulence models in vehicle cabin simulations, based on their suitability for modelling complex airflow patterns and their validation against measured data. It also examines meshing techniques that influence the reliability of CFD simulations, including structured, unstructured, and hybrid meshes. This review article also discusses solver algorithms and their role in efficiently solving the governing equations of fluid flow and heat transfer. Various CFD approaches have been employed to reduce the risk of particle transmission by improving airflow distribution and contaminant removal effectiveness. Moreover, CFD has been used to enhance thermal comfort by optimising temperature distribution and managing heat loads. By synthesising recent advancements and case studies, this review provides valuable insights for researchers and practitioners, aiming to enhance ventilation systems in vehicles through advanced CFD techniques. This work aligns with Sustainable Development Goal (SDG) 3 (Good Health and Well-Being), by improving air quality and reducing infection risks, and SDG 11 (Sustainable Cities and Communities), by contributing to safer and more comfortable transportation solutions. The integration of CFD into vehicle ventilation optimisation supports the creation of healthier and more efficient transportation systems, ultimately contributing to broader sustainability goals.

Keywords: *Thermal comfort, Infection control, Vehicle, Ventilation, Computational fluid dynamics*

DOI: 10.11113/jm.v48.631

1.0 INTRODUCTION

A vehicle is broadly defined as any mechanism designed to transport people or cargo through air, space, water (above or below the surface), or on land [1]. This includes various modes of transport such as automobiles, buses, trains, aircraft, ships, and emerging autonomous or electric vehicles. Each type has its own unique design and operational characteristics that influence indoor environmental quality [2].

Effective vehicle ventilation is crucial for ensuring optimal thermal comfort and maintaining healthy indoor air quality (IAQ) for passengers. Insufficient or poor ventilation systems can lead to uncomfortable conditions, such as temperature imbalances that affect passenger comfort. Furthermore, inadequate ventilation can lead to a higher concentration of airborne pollutants and pathogens, increasing the risk of exposure to harmful substances [3]. This is especially critical in high-occupancy or enclosed settings, where even short-duration trips can expose individuals to elevated concentrations of carbon dioxide (CO_2), bioeffluents, and infectious aerosols.

Besides, this condition is particularly concerning in shared or public transportation environments, where the close proximity of passengers amplifies the potential for the spread of respiratory infections [4] and the buildup of contaminants, making proper ventilation an essential aspect of public health and safety. Urban commuters, school children, healthcare transport patients, and long-distance travellers are among the most vulnerable populations exposed to these risks. The design and management of ventilation systems in these settings, therefore, have direct implications for both comfort and public health.

Figure 1 illustrates the conditions associated with poor thermal comfort and increased pathogen exposure in a bus.

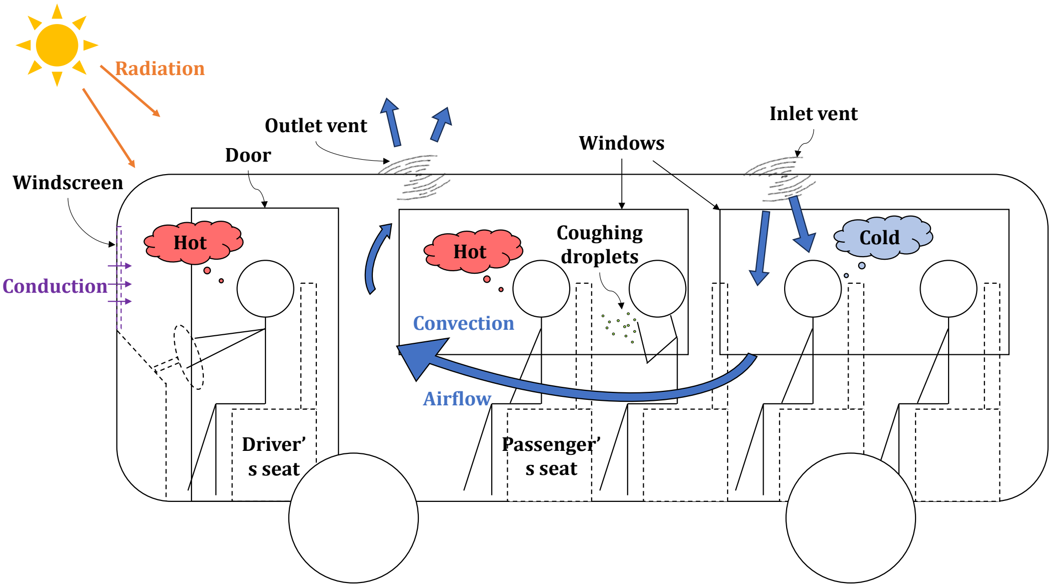

Figure 1: Poor ventilation in a bus leading to thermal discomfort and increased pathogen exposure from a coughing passenger.

According to the ASHRAE-55, thermal comfort significantly influences physical and mental well-being, as individuals prefer environments that align with their comfort needs [5]. The human body generates heat, exchanges it with the environment, and dissipates it through diffusion and the evaporation of body fluids [6]. Even when facing thermal fluctuations, the body's temperature regulation system tries to keep the core body temperature at around 37 °C [6]. Optimising ventilation helps minimise physiological strain by maintaining stable body temperatures and low skin moisture levels, key factors for comfort [7].

In vehicles, the dynamic exposure to solar radiation, varying passenger loads, and fluctuating external temperatures further complicate thermal management [8]. For instance, solar heat gain through transparent surfaces can lead to uneven thermal zones, while idling or slow-moving conditions can reduce the effectiveness of natural airflow, necessitating well-regulated mechanical ventilation.

Considering the studies where individuals spend 1 to 10 hours daily inside vehicles [9], it is important to establish optimal thermal conditions within vehicles. These environments pose unique challenges due to their compact, airtight nature, which makes temperature and humidity regulation critical. In addition, certain vehicle types, such as electric vehicles, limit engine waste heat, further necessitating efficient HVAC systems to maintain passenger comfort and reduce power consumption [10].

The importance of ventilation goes beyond comfort. In the wake of global health crises, such as the COVID-19 pandemic, there has been a heightened focus on airborne transmission of infections within enclosed spaces, including vehicles, where passengers are in close proximity to one another. A study by Fears *et al.* showed that the virus of SARS-CoV-2 can remain infectious in laboratory-generated aerosols for up to 16 hours [11], highlighting the persistence of pathogens in the air. The suspension of pathogens in poorly ventilated environments will increase the risk of transmission among occupants [12], potentially leading to widespread infection.

Moreover, studies on public transport outbreaks [4] have underscored the role of directional airflow and recirculation patterns in facilitating or mitigating cross-infection [13]. This has led to a re-evaluation of ventilation strategies, including the use of HEPA filters, increased fresh air intake, and real-time monitoring of IAQ in transport systems.

Thus, optimising ventilation systems is not only important for regulating temperature but also critical for public health, as it controls transmission of airborne particles. By enhancing vehicle ventilation, we can create safer environments where the risk of infection is minimised, thus protecting public health and ensuring the safety of passengers in shared transportation systems.

Ventilation in vehicles is also tied to safety. Poorly ventilated environments can lead to the buildup of hazardous gases such as CO_2 and volatile organic compounds (VOCs) [14], leading to drowsiness [15] or long-term health effects. For example, inadequate ventilation in long-haul trucks or recreational vehicles (RVs) can compromise driver alertness [16], increasing the risk of accidents. Exposure to elevated CO_2 levels (above 1000 ppm) has been linked to decreased cognitive function and impaired decision-making [17-19], which are critical in driving or emergency response scenarios. Therefore, optimising ventilation systems not only enhances passenger comfort but also plays a crucial role in vehicle safety and health outcomes.

Vast research or review articles have focused on ventilation strategies to enhance thermal comfort and infection control in vehicles, exploring aspects like heat distribution, adjustable air vents, advanced air filtration systems, and airflow patterns. However, review articles specifically addressing the application of Computational Fluid Dynamics (CFD) techniques for modelling these strategies are limited. Existing literature primarily emphasises experimental and simulation approaches, with little attention to the comprehensive use of CFD for optimising vehicle ventilation. This gap in the literature highlights a need for a detailed examination of CFD applications in this context. This review article provides a thorough analysis of how CFD has been used to study and optimise ventilation strategies, focusing on turbulence models, meshing techniques, and solver algorithms. By identifying best practices and integrating considerations for both thermal comfort and infection control, this review will offer valuable insights and recommendations.

This work aligns with Sustainable Development Goal (SDG) 11 (Sustainable Cities and Communities) by contributing to the development of safer, more efficient, and comfortable transportation solutions that enhance the overall quality of urban mobility. It also supports SDG 3 (Good Health and Well-Being) by focusing on the improvement of air quality and the reduction of infection risks, thus promoting healthier environments for both passengers and communities. By encouraging the adoption of simulation-driven design, this review promotes sustainable engineering practices that minimise resource use, shorten development cycles, and enhance resilience in transport systems.

2.0 SYSTEMATIC REVIEW ON CFD TECHNIQUE

CFD is a powerful tool that utilises numerical methods and algorithms to analyse and simulate fluid flow and heat transfer [20, 21]. In vehicle ventilation, CFD plays a crucial role in understanding airflow patterns, temperature distribution, and pollutant dispersion within the vehicle cabin. CFD achieves this by solving the governing equations of fluid motion, including the complex Navier-Stokes equations, which describe the behaviour of viscous fluids. These equations account for various factors such as velocity, pressure, temperature, and turbulence, providing a comprehensive representation of airflow dynamics. One of the major advantages of CFD in vehicle ventilation is its ability to generate detailed visualisations of airflow phenomena [22], which can reveal critical insights that are often difficult to observe through traditional experimental methods.

CFD applications in vehicle ventilation range from optimising air distribution systems to enhancing thermal comfort and minimising the risk of airborne transmission of pathogens. CFD offers a powerful, cost-effective approach for visualising and evaluating airflow, contaminant dispersion, and thermal gradients under various operational conditions, without relying solely on expensive physical prototypes. Through parametric simulations, CFD enables engineers and researchers to evaluate various design scenarios, such as the placement of air vents, the effectiveness of filtration systems, and the impact of different operating conditions on cabin air quality. They can also predict how air moves through the cabin and identify areas of poor circulation. This capability is especially critical in modern vehicles that prioritise energy efficiency and often employ airtight designs, making effective ventilation more challenging.

2.1 Turbulence Models

Turbulence models are essential in CFD simulations as they account for the chaotic and unpredictable nature of fluid flows. Turbulence models in CFD are generally divided into two main categories: Direct Numerical Simulation (DNS) and Reynolds-Averaged Navier-Stokes (RANS) models [23]. Although DNS provides highly detailed and accurate results, its major drawback is the significant computational power it requires, which makes it impractical for larger or more complex geometries [23]. RANS models, while more computationally efficient, sacrifice some accuracy in comparison [23]. To bridge the gap, hybrid models such as Large Eddy Simulation (LES) and Detached Eddy Simulation (DES) have been developed. These models offer a middle ground, providing greater accuracy than RANS while requiring less computational effort than DNS [23].

In CFD, selecting the appropriate turbulence model depends on the flow characteristics. According to Gorman *et al.* [24], for fully laminar flows without any turbulence, a laminar solver is sufficient. However, for fully turbulent flows, particularly those bounded by walls, the SST (Shear Stress Transport) model is highly recommended due to its ability to capture complex flow phenomena and its reliable performance in thermal-transport scenarios [24]. When dealing with flows that are a mix of laminar, transitional, or turbulent, or when flows are time-dependent (such as pulsatile flows), the

SST transitional model is preferred [24]. For situations requiring the capture of small-scale, transient flow phenomena, models like the Scale-Adaptive SST or LES are suggested [24].

Several turbulence models commonly employed in vehicle ventilation studies are shown in Table 1 below.

Table 1: Comparison of commonly employed turbulence models in vehicle studies [24, 25]

Turbulence model	Pros	Cons
k-ε model	• Simple formulation. • Strong performance across applications. • Extensively validated in various scenarios. • Preferred for its computational efficiency.	• Struggles with capturing complex flow behaviours, less accurate in cases with strong separation or swirling flows. • Challenges in boundary layer zones. • Performance issues at low Reynolds numbers.
k-ω model	• More accurate for boundary layer flows. • Performs well in low Reynolds number conditions. • Ideal for detailed airflow studies near vehicle surfaces.	Performance dependent on free-stream turbulence parameters.
LES	• Resolves large-scale turbulent structures. • Models smaller scales efficiently. • Offers improved accuracy for transient flow behaviour.	Computationally intensive.

In addition, the standard k-ε turbulence model [26, 27] and the re-normalisation group (RNG) k-ε turbulence model [28, 29] are widely used for simulating the turbulent flow field inside a chamber. To highlight, RNG k-ε is popular modification of the traditional k-ε model, offering better performance than the standard k-ε, particularly for small-scale and rotating flows [24].

When selecting a turbulence model for vehicle ventilation scenarios, factors such as the geometry of the vehicle cabin, the expected flow patterns, and the computational resources available must be considered. Case studies have demonstrated the effectiveness of these models in optimising ventilation systems. For instance, simulations by Tan *et al.* [30] using the k-ε model identified optimal airflow velocity and temperature distribution in an aircraft cabin, meeting airworthiness standards for human comfort.

2.2 Meshing Techniques in CFD

2.2.1 Structured, Unstructured and Hybrid Mesh

In CFD, mesh serves as the foundation for numerical simulations, discretising the physical domain into smaller elements that allow for the solution of governing equations. The choice between structured and unstructured meshes significantly affects the accuracy, efficiency, and applicability of the simulation [31]. Figure 2 illustrates structured and unstructured meshes applied to a simple model.

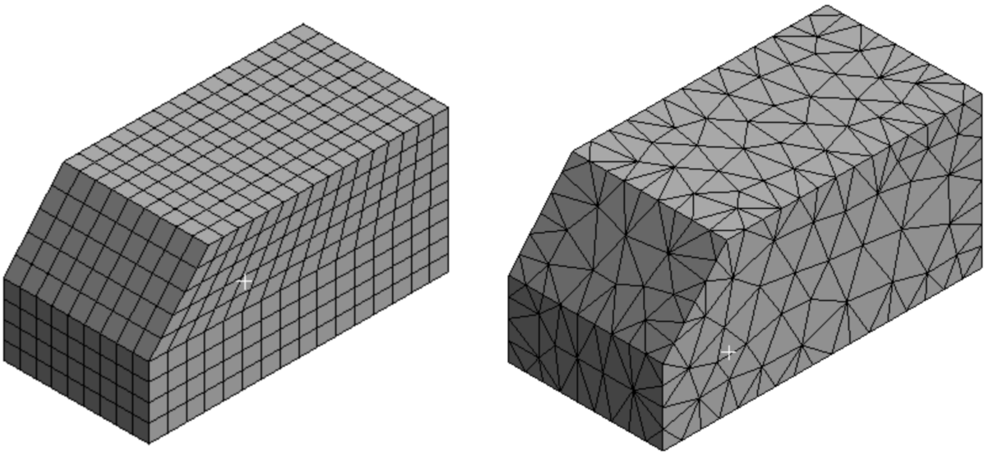

Figure 2: Two basic mesh types in CFD simulations: (a) structured mesh and (b) unstructured mesh.

Structured meshes are characterised by a regular, grid-like arrangement of cells, typically using quadrilateral cells in 2D or hexahedral cells in 3D [32]. The structured nature of these meshes allows for a high degree of control over the alignment of the mesh with the flow direction, which can be particularly advantageous in scenarios with simple, predictable geometries. The regularity of structured meshes facilitates the use of efficient numerical algorithms, leading to faster computation times and lower memory usage [32, 33]. This makes structured meshes highly effective for simulations where the geometry aligns well with the mesh, such as airflow over flat surfaces or within simple ductwork systems. However, the primary limitation of structured meshes is their lack of flexibility in handling complex geometries [33]. In vehicle environments, simulating the airflow distribution around the contoured seats and headrests of passengers with a structured mesh can result in a loss of accuracy or a significant increase in computational resources due to the need for fine meshing. Due to these limitations, structured meshes are not widely used in vehicle airflow simulations where complex geometries are present, as they can become inefficient and less practical for capturing the detailed flow phenomena necessary for optimising passenger comfort and air distribution within the cabin.

Unstructured meshes consist of irregularly shaped cells, such as triangles in 2D or tetrahedra in 3D [32]. This irregularity allows unstructured meshes to easily conform to complex geometries [32, 34], making them well-suited for CFD simulations in environments with intricate shapes and details. The flexibility of unstructured meshes is particularly advantageous in simulations where the geometry is complex [35], such as the intricate surfaces of a vehicle’s interior or the detailed contours of a vent [36]. Unstructured meshes can adapt to these complex shapes without requiring the extensive refinement that structured meshes might need [37], allowing for more accurate modelling of airflow and thermal distribution in challenging areas. Despite their flexibility, unstructured meshes are more computationally demanding than structured meshes [33] as the irregularity of the cells requiring more sophisticated algorithms and higher memory usage [32]. Unstructured meshes may introduce numerical errors, particularly in regions where flow alignment is critical. For instance, in areas where precise boundary layer resolution is needed, such as near the surfaces of seats or passengers. Unstructured meshes may struggle to provide the same level of accuracy as structured meshes, potentially affecting the reliability of predictions in those regions.

In this context, hybrid meshes combine the advantages of both structured and unstructured meshes by integrating structured elements in regions of simple geometry and unstructured elements in more complex areas. This approach allows for optimised meshing that balances computational efficiency with the need for flexibility in handling complex shapes [38]. However, the complexity of generating and managing hybrid meshes with high aspect ratio, especially ensuring smooth transitions between different mesh types, poses significant challenges [37]. This complexity can lead to numerical errors if not carefully handled, and the additional computational overhead may reduce the anticipated efficiency gains. As a result, despite their potential benefits, hybrid meshes are less commonly used in airflow and contaminant studies, where simpler meshing approaches often suffice and are more straightforward to implement.

2.2.2 Mesh Refinement

Mesh refinement is critical for ensuring that CFD simulations accurately capture important flow features, especially in regions with high gradients or complex flow behaviour [39]. In vehicle ventilation simulations, refined meshes are necessary to resolve details like boundary layers around passengers, the intricate flow patterns around vents, and the distribution of particles or contaminants within the cabin.

Researchers apply various techniques to achieve effective mesh refinement. Adaptive mesh refinement is one the popular method, where the mesh is dynamically refined in regions with significant flow changes during the simulation [31]. This allows for higher resolution where it is most needed, without unnecessarily increasing the computational burden across the entire domain. Another key method is the grid independence test. This involves running simulations with progressively finer meshes and comparing the results to ensure that further refinement does not lead to significant changes in the outcome. Achieving grid independence is crucial as it confirms that the solution is no longer dependent on the mesh size, indicating that the results are accurate and reliable. Baker *et al.* [40] emphasises the importance of the Grid Convergence Index (GCI) method, which requires at least three systematic mesh refinements to ensure that the numerical solution is in the asymptotic range. GCI is a quantitative measure that provides a relative error bound to describe how a numerical solution varies with successive mesh refinements [41]. It is designed to assess the convergence of a solution as the mesh is refined, helping to determine whether the results are approaching the true physical solution. A lower the GCI value indicates a higher level of accuracy, and it could be achieved when the observed order of accuracy closely aligns with the formal order of accuracy. Researchers applied a GCI value of 5% to 10% to ensure grid independency when performing indoor airflow studies [42, 43].

Maintaining high mesh quality is essential for obtaining accurate CFD results, as the mesh forms the foundation for the entire simulation. A well-constructed mesh ensures that the computational model can accurately capture the physical phenomena of fluid flow, heat transfer, and pollutant dispersion. Poor mesh quality, characterised by excessively skewed or distorted cells, can introduce numerical errors, such as artificial diffusion or incorrect flow behaviour, which can degrade the accuracy of the simulation [44]. This is particularly important in vehicle CFD simulations, where precise control of airflow and particle dispersion is crucial for optimising thermal comfort and infection control.

2.2.3 Solver Algorithms and Techniques

Solver algorithms are crucial in CFD to ensure the reliability of predicting fluid flow and heat transfer within vehicle cabins [45]. Among the widely used solver algorithms are Semi-Implicit Method for Pressure-Linked Equations (SIMPLE), Semi-Implicit Method for Pressure-Linked Equations-Consistent (SIMPLEC), Semi-Implicit Method for Pressure-Linked Equations Revised (SIMPLER), and Pressure-Implicit with Splitting of Operators (PISO) [46], as presented in Figure 3. These algorithms play a significant role in simulating indoor vehicle environments, where precise modelling of airflow, temperature distribution, and contaminant dispersion is essential [15].

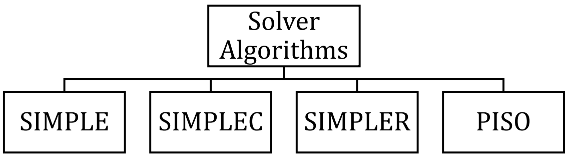

Figure 3: Four widely used solver algorithms in CFD simulations.

The SIMPLE algorithm is commonly used to solve the Navier-Stokes equations in incompressible flows [47]. It applies to most indoor airflow cases, such as vehicle cabins. This algorithm operates by iterating between the velocity and pressure fields to ensure mass conservation. The algorithm involves a predictor step, where an intermediate velocity field is estimated using the previous pressure field. This is followed by a pressure correction step, which adjusts the pressure field to satisfy the continuity equation. While SIMPLEC and SIMPLER are the variant of the SIMPLE algorithm, and widely used for indoor simulation [39]. SIMPLEC algorithm enhances the convergence speed by optimising the pressure correction equation and reduce under-relaxation factor, while SIMPLER algorithm enhances the convergence characteristics by using integrating additional pressure corrections step [48]. In the context of vehicle cabin simulations, SIMPLE is appropriate for predicting airflow patterns and ventilation performance. Its robustness and good convergence properties make it suitable for steady-state problems, although it may require more iterations and computational resources compared to other methods.

The PISO algorithm extends the SIMPLE method to improve transient flow simulations. It is useful in scenarios where there are rapid changes in flow conditions, such as airflow induced by human movement within a vehicle or sneezing and coughing by occupants [49]. The PISO algorithm incorporates additional steps to enhance accuracy and stability in transient simulations. It involves a pressure correction step [50] similar to SIMPLE, but with extra iterations to refine the pressure field. A velocity update step follows, ensuring continuity and updating the velocity field. A second pressure correction step [50] further refines the pressure field, offering improved accuracy for time-dependent problems. Although PISO offers better performance for transient simulations, it is more computationally demanding due to its additional iterations and steps [50].

In comparing the two algorithms, SIMPLE is generally favoured for steady-state simulations involving complex geometries, due to its robustness and convergence properties. However, for dynamic conditions and time-dependent simulations, such as varying ventilation rates and human movement, PISO offers improved accuracy and stability [51]. A recent study revealed that the Extension of SIMPLE (ESIMPLE) algorithm demonstrates a superior convergence rate compared to the traditional SIMPLE, PISO, and SIMPLEC algorithms [52]. Understanding the specific requirements of the simulation could aid in selecting the appropriate algorithm to achieve reliable and accurate simulation results. Table 2 tabulates the combinations of turbulent model, conditions, and algorithms used in previous vehicle cabin studies.

Table 2: Simulation condition and algorithm used in the past vehicle cabin studies

Reference	Main Parameter	Turbulent Model		Condition		Algorithm				
		RANS	Non-RANS (LES, DES, etc)	Steady (Time-independent)	Transient (Time dependent)	SIMPLE	SIMPLEC	PISO	COUPLED	Not Applicable
[53]	Airflow distribution	✓		✓		✓				
[54]	Particle dispersion	✓		✓		✓				
[55]	Particle dispersion		✓		✓					✓
[45]	Airflow distribution	✓		✓		✓				
[56]	Air contaminant	✓		✓			✓			
[57]	Air contaminant	✓		✓	✓	✓ (steady)				✓ (transient)
[58]	Temperature distribution	✓			✓				✓	
[59]	Temperature distribution	✓		✓		✓				

3.0 CFD APPLICATIONS IN VEHICLE VENTILATION

Several studies of such and similar problems to passenger car model highlight the use of the RNG k-ε model [60, 61], while some support the use of SST k-ω for similar applications [62, 63]. Christian Suárez further supports the SST k-ω model's suitability for steady-state airflow simulations, treating air as an ideal gas in vehicle environments.

3.1 Airflow Patterns

Airflow patterns refer to the air velocity distributed throughout the vehicle, which may be varies based on a few critical factors, as summarised in Figure 4.

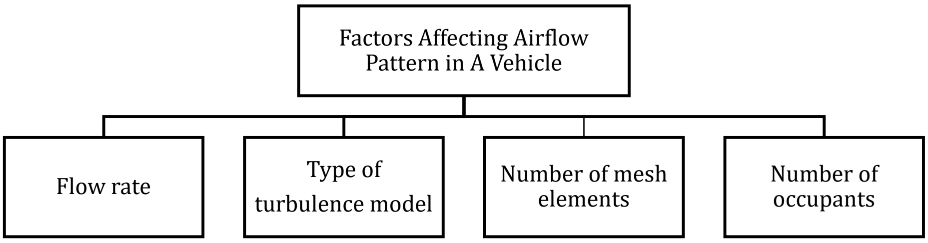

Figure 4: Summary of critical factors affecting airflow pattern in a vehicle.

The first factor is the flow rate, Q (m^3/s) throughout the vehicle. Danca *et al.* [64] conducted an CFD simulation for a sedan vehicle using turbulence model of RNG k-ε with 6.5 million tetrahedral mesh elements. The study showed that when the boundary conditions for the central, right and left diffuser had Q of 0.0131 m^3/s, 0.0068 m^3/s and 0.0047 m^3/s respectively, the maximum air velocity experienced by the driver and front passenger at x = 1.35 m was 0.80 m/s, while the back passengers at x = 2.8 m experienced a maximum velocity of 0.30 m/s, with a maximum air velocity of 1.00 m/s from each inlet [64]. In comparison, with boundary conditions at higher flow rate, where the central, right and left diffuser were set to 0.0380 m^3/s, 0.0197 m^3/s and 0.0158 m^3/s respectively, the maximum velocity experienced by the front passenger at x = 1.35 m was 1.4 m/s, and by back passenger at x = 2.8 m was 1.1 m/s, with a maximum air velocity of 2 m/s at inlet [64]. Fišer & Pokorny [65] reported that fan speed impacts flow rate. At fan speed 2, Q was 0.02 m^3/s with a maximum inlet air velocity of 2.6 m/s, and at fan speed 4, Q increased to 0.05 m^3/s with a maximum inlet velocity of 5.8 m/s [65].

Besides, the factor affecting airflow patterns include the type of turbulence model and the number of mesh elements used in the simulation. In terms of turbulence model, Djeddou *et al.* [66] conducted a study to compare airflow velocity between SST k-ω and RNG k-ε model using 28 million in a sedan car. They reported that at the point of x = 1.2 m, the airflow velocity using the RNG k-ε model was 0.4 m/s, while the SST k-ω model gave an airflow velocity of 0.9 m/s [66]. In terms of mesh element, Djeddou *et al.* [54] conducted another a CFD simulation on a sedan car using SST k-ω turbulence model with 28 million tetrahedral mesh elements, resulting in airflow velocity of 0.6 m/s at x = 1.2 m. When using 11 million tetrahedral mesh elements, the airflow velocity at the same point was 0.5 m/s [54].

Moreover, the size of the vehicle is one of the factors affecting airflow patterns in vehicle cabin. Khatoon & Kim [67] conducted simulation on a vehicle with interior dimension of 1.8 m (L) × 1.5 m (W) × 1.0 m (H), using RNG k-ε turbulence model with 3.2 million tetrahedral mesh elements. In the study, the maximum air velocity from the inlet is 0.463 m/s, and the maximum air velocity experienced by passengers at the point of x = 1.4 m is 0.30 m/s [67]. To compare, another study by Pirouz *et al.* [68] used larger car with dimension of 2.5 m (L) × 1.2 m (W) × 1.2 m (H). From their study, they reported that when Q = 0.06 m^3/s, maximum airflow velocity experienced by front passenger at the point of x = 1.2 m is 0.71 m/s.

The number of occupants in a car is also a critical factor affecting the airflow velocity. Djeddou *et al.* [54] conducted CFD simulation using SST k-ω model with 28 million tetrahedral mesh elements to compare airflow velocity between one and four occupants in a sedan. They reported that with one occupant, the maximum air velocity at x = 1.2 m was 0.9 m/s, and at x = 2.4 m, it was 0.8 m/s [54]. In comparison, with four occupants, the maximum air velocity at x = 1.2 m dropped to 0.8 m/s, and at x = 2.4 m, it decreased to 0.5 m/s due to the obstruction of airflow by human manikin in the front passenger seat [54].

To add on, the number of opened windows when the car is moving also influence the airflow patterns in the vehicle. Mathai *et al.* [69] performed a study where the car was moving at 80 km/h with different number of opened windows. The study reported that when all four windows were opened, the ventilation airflow rate is 50 L/s; when only two windows are opened, which were the rear-left and front-right window, the ventilation airflow rate is 30 L/s [69].

3.2 Thermal Comfort

In the highly asymmetric and transient environment of an automobile passenger cabin, the air temperature is only one of many key environmental factors that affect thermal comfort [70]. Air temperature within vehicles often exhibits non-uniformity due to the installed air conditioning system [71]. The impact of air movement on human comfort is significant, especially in sensitive areas like the neck, head, and feet, varying based on individual sensitivity. Excessive or irregular airflow can lead to local thermal discomfort [72].

ASHRAE-55, which outlines thermal environmental conditions for human occupancy, states that the temperature in a vehicle cabin should be between 20 °C and 25 °C to ensure a good thermal comfort of the passenger in a car cabin. AHSRAE thermal sensation scale introduced Predicted Mean Voted (PMV), representing the average thermal sensation felt by a group [73]. PMV value is divided into seven values, ranges from "–3" (Cold), "–2" (Cool), "–1" (Slightly Cool), "0" (Neutral), "+1" (Slightly Warm), "+2" (Warm) to "+3" (Hot) [73]. ASHRAE-55 defines a comfort zone within PMV levels from "–0.5" to "+0.5" [74]. In other words, the nearer the PMV value to 0 (Neutral), the better the thermal comfort experienced by the occupants. To complement PMV, Predicted Percentage of Dissatisfied (PPD) is introduced. PPD calculates the prediction of the number of thermally dissatisfied individuals within a group [75].

Karthick *et al.* [76]'s study investigated thermal comfort in an SUV cabin using CFD simulations with the k-ε turbulence model. Karthick noted that cabin temperature depends on factors such as cabin size, the number and form of air vents, HVAC system mass flow rate, and interior materials like the dashboard and seats. The study examined various vehicle speeds to find a suitable temperature, with airflow simulations modelling an inlet velocity of 10 m/s, resulting in a cabin temperature of 23 °C and airflow velocity of 5 m/s. Dynamic air vents were recommended to improve airflow and HVAC efficiency [76].

Next, Taftian [77] conducted a study in a recreational vehicle (RV), with the RNG k-ε turbulence model to solve transient 3D RANS equations and applied the SIMPLE algorithm for pressure-velocity coupling. In the study, several parametric studies had been done to optimise vent configurations, by analysing inlet and outlet locations, airflow direction, and the number of exhaust vents using metrics like temperature, velocity, PMV, and PPD values. He reported that vertical supply air improved mixing, while two exhaust vents significantly reduced hot spots and enhanced thermal comfort [77]. The optimised configuration outperformed the reference case, showing better airflow uniformity and temperature distribution [77], which is important for better thermal comfort of occupants in RV.

Cigarini *et al.* [78] experimentally investigated the impact of HVAC systems on energy consumption and thermal comfort in a 12-meter Battery Electric Bus (e-bus). The study employed a custom-developed sensor station to measure climatic parameters and identified a suitable thermal comfort model. Data on energy consumption, battery State of Charge (SoC), and available travel range were recorded using an embedded data logger. Tests were conducted under winter conditions on a Berlin bus line, comparing scenarios with the heating system on and off. The study reported that heating increased the e-bus's energy consumption by a factor of 1.9, significantly reducing the battery's SoC and available travel range [78]. Thermal comfort was evaluated as shifting from "comfortable" with heating on to "slightly uncomfortable but acceptable" with heating off [78].

Jose and Chidambaram [79] conducted a study on thermal comfort in a minivan cabin using CFD simulations with the RNG k-ε turbulence model and validated their findings with experimental data. The study analysed air velocity and temperature, observing that air velocity inside the vehicle was set to 2.3 m/s with a temperature of 23 °C, close to ASHRAE standards. The highest PPD values were recorded at the walls and AC vent outlets, with comfort indices remaining within the neutral PMV scale range for optimised AC positions, enhancing passenger comfort across different scenarios [79].

3.3 Infection Control

Vehicle ventilation is crucial in reducing airborne particle concentrations that could pose health risks to passengers. According ASHRAE-62.1, which outlines ventilation for acceptable indoor air quality and can be used for designing vehicle HVAC systems, the ideal air velocity in a car should be between 0.2 m/s and 1 m/s. This range helps prevent turbulent airflow that can hinder the removal of airborne particle through exhaust.

ASHRAE-62.1 also specifies that the relative humidity in a vehicle should be maintained between 30% and 60%, with an ACH of 20 to 60, due to the smaller size of the space.

A particle is composed of three components, which are dust, allergens and pathogens. Sattar *et al.* [80] found that in a vehicle cabin, particles consist of 40-60% dust, 20-30% allergens, and 5-15% pathogens. To further reduce airborne concentrations in cars, Mariita *et al.* [81] suggested mounting Ultraviolet (UV) light lamps on HVAC systems. They reported a reduction of airborne concentration up to 95.14% in the vehicle cabin after 5 minutes of sampling, compared to without UV light. The HVAC system, using RNG k-ε turbulence model, achieved an efficiency of 82.34% in removing airborne concentrations during the same time period [81]. Another study by Pan *et al.* [55] reported that the installation of Ultraviolet C (UVC) lamps with a wavelength of 222 nm further reduces the normalised total number of bioaerosols inhaled by the driver to 1.19×10^{-7}, representing an additional 32.1% reduction. This follows the installation of a partition, which decreases aerosol inhalation by 55.9%, equivalent to a reduction in the normalised total number of bioaerosols to 2.46×10^{-7} at $t = 6$ s.

Pan *et al.* also [55] conducted a study using the LES method and reported that opening the car window reduced the normalised total number of bioaerosol inhaled by driver from 3.38×10^{-5} to 8.83×10^{-7}, corresponding to a reduction of 97.4%. Feng *et al.* [82] found that one ion generator in the car's HVAC system achieved 35% of disinfection efficiency, and increasing to 55% with two ion generators. They also reported that using an Electrostatic Disinfector (ESD) at 6.5 kV and an Enhanced Air Filtration System (EEAF), CFD results showed approximately 100% filtration efficiency for particles sizing between 0.1 and 2.5 μm, while saving up to 87.9% in car fuel energy consumption [82]. Wang *et al.* [83] suggested the use of High-Efficiency-Particulate-Filter (HEPA), which has an efficiency of up to 99.97% at removing dust, pollen, mould, bacteria, and any airborne particles with a size of 0.3 μm or larger.

4.0 CONCLUSION

The choice of turbulence model significantly impacts the accuracy and efficiency of CFD simulations in vehicle ventilation scenarios. Case studies, such as those by Djeddou *et al.* [66], Jose and Chidambaram [79] and Mariita *et al.* [81] demonstrate that models like the RNG k-ε and SST k-ω can effectively optimise airflow and temperature distribution, ensuring compliance with human comfort standards while considering vehicle geometry and computational resources. Additionally, the selection of meshing techniques and solver algorithms is critical for accurately simulating airflow and thermal dynamics within vehicle cabins; structured, unstructured, and hybrid meshes offer distinct advantages, while algorithms like SIMPLE and PISO enhance the reliability of CFD predictions.

The effectiveness of CFD simulations in vehicle ventilation is underscored by their ability to analyse airflow patterns, thermal comfort, and infection control. CFD provides valuable insights into the complex interactions between air movement, temperature distribution, and the dispersion of particles, which are essential for designing efficient ventilation systems. The studies reviewed emphasise that optimising airflow dynamics within vehicle cabins is fundamental for achieving thermal comfort and improving IAQ. Key to this optimisation is the integration of advanced turbulence models, which better represent the chaotic nature of airflow in confined spaces. In addition, the implementation of innovative technologies, such as UV lamps for pathogen inactivation and HEPA filters for fine particulate matter removal, has been shown to significantly enhance air quality and reduce the risk of infection transmission. These technologies, in combination with well-calibrated ventilation strategies, play a critical role in creating healthier and more comfortable environments for passengers, especially in shared or high-traffic modes of transportation.

For future work and applications, researchers should focus on refining turbulence models and meshing techniques to enhance the accuracy of CFD simulations in vehicle ventilation, as these choices significantly influence results. Enhancing the precision of these models will help in understanding complex interactions and improving the overall design of ventilation systems. Moreover, there is a need for further investigations into integrating advanced technologies, in simulation studies to better understand their impact on air quality and thermal comfort in vehicle cabins. On the other hand, practitioners should leverage these enhanced simulation insights to inform the design and optimisation of vehicle ventilation systems. By adopting evidence-based strategies, they can develop more effective ventilation solutions that prioritise passenger comfort, reduce the risk of airborne illnesses, and ensure overall safety in the shared transportation systems.

ACKNOWLEDGEMENT

The authors would like to acknowledge the financial support from Asia Technological University (ATU) Network under ATU-Net Young Researcher Grant (YRG) with Vot. No. R.J130000.7724.4J714 and Universiti Teknologi Malaysia under UTM Nexus Postgrad Grant with Vot. No. Q.J130000.5324.00L96.

REFERENCES

1. Liao, Y.G., Vehicles: A New Open Access Journal to Publish Your Ground Vehicle Research Results. Vehicles, 2019. 1: p. 1-2.
2. Tran, P.T.M., et al., In-car occupants' exposure to airborne fine particles under different ventilation settings: Practical implications. Atmospheric Environment, 2024. 318: p. 120271.
3. Deol, A.K., et al., Estimating ventilation rates in rooms with varying occupancy levels: Relevance for reducing transmission risk of airborne pathogens. PLoS One, 2021. 16(6): p. e0253096.
4. Helfers, A., et al., Visualizing ventilation in the bus: Addressing risk perception in public transport passengers. Transportation Research Part F: Traffic Psychology and Behaviour, 2024. 101: p. 236-249.
5. American Society of Heating, Refrigerating, and Air Conditioning Engineers. Thermal environment conditions for human occupancy: ANSI/ASHRAE Standard 55-2013. 2013; Available from: https://www.ashrae.org/file%20library/technical%20resources/standards%20and%20guidelines/standards%20addenda/55_2013_b_20141209.pdf.
6. Hoof, J.v., M. Mazej, and J.L.M. Hensen, Thermal comfort: research and practice. FBL, 2010. 15(2): p. 765-788.
7. Djongyang, N., R. Tchinda, and D. Njomo, Thermal comfort: A review paper. Renewable and Sustainable Energy Reviews, 2010. 14(9): p. 2626-2640.
8. Guo, N., et al., Dynamic thermal radiation regulation for thermal management. Next Energy, 2023. 1(4): p. 100072.
9. Frank, L.D., M.A. Andresen, and T.L. Schmid, Obesity relationships with community design, physical activity, and time spent in cars. American Journal of Preventive Medicine, 2004. 27(2): p. 87-96.
10. Lajunen, A., Y. Yang, and A. Emadi, Review of Cabin Thermal Management for Electrified Passenger Vehicles. IEEE Transactions on Vehicular Technology, 2020. 69(6): p. 6025-6040.
11. Fears, A.C., et al., Persistence of Severe Acute Respiratory Syndrome Coronavirus 2 in Aerosol Suspensions. Emerg Infect Dis, 2020. 26(9): p. 2168-71.
12. Morawska, L., et al., How can airborne transmission of COVID-19 indoors be minimised? Environment International, 2020. 142: p. 105832.
13. Nielsen, P.V. and C. Xu, Multiple airflow patterns in human microenvironment and the influence on short-distance airborne cross-infection – A review. Indoor and Built Environment, 2022. 31(5): p. 1161-1175.
14. Sørensen, S.B. and K. Kristensen, Low-cost sensor-based investigation of CO2 and volatile organic compounds in classrooms: Exploring dynamics, ventilation effects and perceived air quality relations. Building and Environment, 2024. 254: p. 111369.
15. Chang, T.B., et al., Development of a CFD model for simulating vehicle cabin indoor air quality. Transportation Research Part D: Transport and Environment, 2018. 62: p. 433-440.

16. Marshall, G.J., et al., Thermal Management of Vehicle Cabins, External Surfaces, and Onboard Electronics: An Overview. Engineering, 2019. 5(5): p. 954-969.
17. Satish, U., et al., Is CO2 an indoor pollutant? Direct effects of low-to-moderate CO2 concentrations on human decision-making performance. Environ Health Perspect, 2012. 120(12): p. 1671-7.
18. Ezeamii, V., et al., Air Quality Monitoring in Schools: Evaluating the Effects of Ventilation Improvements on Cognitive Performance and Childhood Asthma. Cureus, 2025.
19. Fan, Y., et al., Short-term exposure to indoor carbon dioxide and cognitive task performance: A systematic review and meta-analysis. Building and Environment, 2023. 237: p. 110331.
20. Mund, C., S.K. Rathore, and R.K. Sahoo, A review of solar air collectors about various modifications for performance enhancement. Solar Energy, 2021. 228: p. 140-167.
21. Atoyebi, E.O., et al., 2.05 - Scaffold modeling advancement in biomaterials application, in Comprehensive Materials Processing S. Hashmi, Editor. 2024, Elsevier: Oxford. p. 56-71.
22. Hafiz, M., et al. A review: Fundamentals of computational fluid dynamics (CFD). in Proceedings of the 4th International Conference on Green Design and Manufacture. 2018. Ho Chi Minh.
23. Hami, K., Turbulence Modeling a Review for Different Used Methods. International Journal of Heat and Technology, 2021. 39: p. 227-234.
24. Gorman, J., et al., Turbulence Models Commonly Used in CFD, in Applications of Computational Fluid Dynamics Simulation and Modeling, S. Bhattacharyya, Editor. 2021, IntechOpen: Rijeka.
25. Halim, M., et al., The Evaluation of k-ε and k-ω Turbulence Models in Modelling Flows and Performance of S-shaped Diffuser. International Journal of Automotive and Mechanical Engineering, 2018. 15: p. 5161-5177.
26. Zhang, H., et al., Studies of air-flow and temperature fields inside a passenger compartment for improving thermal comfort and saving energy. Part II: Simulation results and discussion. Applied Thermal Engineering, 2009. 29(10): p. 2028-2036.
27. Zhang, H., et al., Studies of air-flow and temperature fields inside a passenger compartment for improving thermal comfort and saving energy. Part I: Test/numerical model and validation. Applied Thermal Engineering, 2009. 29(10): p. 2022-2027.
28. Mao, Y., J. Wang, and J. Li, Experimental and numerical study of air flow and temperature variations in an electric vehicle cabin during cooling and heating. Applied Thermal Engineering, 2018. 137: p. 356-367.
29. Bianco, V., et al., Numerical investigation of transient thermal and fluidynamic fields in an executive aircraft cabin. Applied Thermal Engineering, 2009. 29(16): p. 3418-3425.
30. Tan, J., et al., Numerical Simulation of Airflow Velocity and Temperature Distribution in an Aircraft Cabin. Journal of Physics: Conference Series, 2024. 2747(1): p. 012042.
31. Karkoulias, D.G., et al., Computational Fluid Dynamics Study of Wing in Air Flow and Air–Solid Flow Using Three Different Meshing Techniques and Comparison with Experimental Results in Wind Tunnel. Computation, 2022. 10(3): p. 34.
32. Allison, C. Meshing in FEA: Structured vs Unstructured meshes. 2020; Available from: https://onscale.com/meshing-in-fea-structured-vs-unstructured-meshes/?utm_source=chatgpt.com.
33. Ali, Z., P.G. Tucker, and S. Shahpar, Optimal mesh topology generation for CFD. Computer Methods in Applied Mechanics and Engineering, 2017. 317: p. 431-457.
34. Wijesooriya, K., et al., A technical review of computational fluid dynamics (CFD) applications on wind design of tall buildings and structures: Past, present and future. J. Build. Eng., 2023. 74: p. 106828.
35. Wijesooriya, K., et al., A technical review of computational fluid dynamics (CFD) applications on wind design of tall buildings and structures: Past, present and future. Journal of Building Engineering, 2023. 74: p. 106828.
36. Bode, F., et al., Impact of realistic boundary conditions on CFD simulations: A case study of vehicle ventilation. Building and Environment, 2025. 267: p. 112264.
37. Li, H., L. Rong, and G. Zhang, Reliability of turbulence models and mesh types for CFD simulations of a mechanically ventilated pig house containing animals. Biosystems Engineering, 2017. 161: p. 37-52.
38. Kaya, K. and O. Özcan, A numerical investigation on aerodynamic characteristics of an air-cushion vehicle. Journal of Wind Engineering and Industrial Aerodynamics, 2013. 120: p. 70-80.
39. Tan, H., et al., Utilising localised exhaust and air curtain to reduce airborne particle settlement on surgical patients: potential future application in operating rooms? Journal of Thermal Analysis and Calorimetry, 2024. 149: p. 11323–11336.
40. Baker, N., G. Kelly, and P. O'Sullivan, A grid convergence index study of mesh style effect on the accuracy of the numerical results for an indoor airflow profile. International Journal of Ventilation, 2020. 19(4): p. 300-314.
41. Wang, S., et al., Integrated Predictions of the Influence of Mesh Size, Casting Defects and SDAS on the Fatigue Life of Aluminum Alloy Wheels. Journal of Materials Research and Technology, 2025. 35: p. 3956-3967.
42. Kek, H.Y., et al., Is thermal-guided mobile air supply a practical measure in burn isolation wards? Potential future applications. Journal of Thermal Analysis and Calorimetry, 2024.
43. Volk, A., U. Ghia, and C. Stoltz, Effect of grid type and refinement method on CFD-DEM solution trend with grid size. Powder Technology, 2017. 311: p. 137-146.

44. Katz, A. and V. Sankaran, Mesh quality effects on the accuracy of CFD solutions on unstructured meshes. Journal of Computational Physics, 2011. 230(20): p. 7670-7686.
45. Tan, L. and Y. Yuan, Computational fluid dynamics simulation and performance optimization of an electrical vehicle Air-conditioning system. Alexandria Engineering Journal, 2022. 61(1): p. 315-328.
46. Tan, H., et al., Current and potential approaches on assessing airflow and particle dispersion in healthcare facilities: a systematic review. Environ Sci Pollut Res Int, 2022. 29(53): p. 80137-80160.
47. Zienkiewicz, O.C., R.L. Taylor, and P. Nithiarasu, Chapter 1 - Introduction to the Equations of Fluid Dynamics and the Finite Element Approximation, in The Finite Element Method for Fluid Dynamics O.C. Zienkiewicz, R.L. Taylor, and P. Nithiarasu, Editors. 2014, Butterworth-Heinemann: Oxford. p. 1-29.
48. Liu, X.L., W.Q. Tao, and Y.L. He, A simple method for improving the SIMPLER algorithm for numerical simulations of incompressible fluid flow and heat transfer problems. Engineering Computations, 2005. 22(8): p. 921-939.
49. Tan, H., et al., Does human movement-induced airflow elevate infection risk in burn patient's isolation ward? A validated dynamics numerical simulation approach. Energy and Buildings, 2023. 283: p. 112810.
50. Zhou, P., et al., Performance evaluation of different pressure-velocity decoupling schemes in built environment simulation. Energy and Buildings, 2022. 257: p. 111763.
51. Kek, H.Y., et al., Evaluating the impact of human movement-induced airflow on particle dispersion: A novel real-time validation using IoT technology. Energy and Buildings, 2024. 323: p. 114825.
52. Serra, N. and V. Semiao, ESIMPLE, a new pressure–velocity coupling algorithm for built-environment CFD simulations. Building and Environment, 2021. 204: p. 108170.
53. Boughanmi, N., J. Frisch, and C. van Treeck, Evaluation of the airflow distribution of a vertical air ventilation system in a car cabin using PIV measurements and CFD simulations. International Journal of Heat and Fluid Flow, 2024. 107: p. 109411.
54. Djeddou, M., et al., A Diffusion-Inertia Model for the simulation of particulate pollutants dynamics inside a car cabin. Journal of Aerosol Science, 2024. 175: p. 106279.
55. Pan, Y., et al., Evaluation of intervention measures in reducing the driver's exposure to respiratory particles in a taxi with infected passengers. Science of The Total Environment, 2023. 902: p. 166099.
56.Qin, D., et al., Indoor air formaldehyde (HCHO) pollution of urban coach cabins. Scientific Reports, 2020. 10: p. 332.
57. Chang, T.-B., et al., Development of a CFD model for simulating vehicle cabin indoor air quality. Transportation Research Part D: Transport and Environment, 2018. 62: p. 433-440.
58. Bode, F., et al. The influence of the Inlet angle of vehicle air diffuser on the thermal comfort of passengers. in 2017 Int. Conf. Energy Environ. (ICEM) 2017. Bucharest.
59. Zeng, Y., M. Katsuta, and T. Anamizu, Numerical Simulation of Heat Flow in a Vehicle Cabin Using the Personal Air Conditioning System. Advanced Materials Research, 2013. 694-697: p. 755-761.
60. Kılıç, M. and G. Sevilgen, The effects of using different type of inlet vents on the thermal characteristics of the automobile cabin and the human body during cooling period. International Journal of Advanced Manufacturing Technology - INT J ADV MANUF TECHNOL, 2012. 60: p. 799–809.
61. Mazej, M. and V. Butala, Investigation in the Characteristics of the Personal Ventilation Using Computational Fluid Dynamics. Indoor and Built Environment, 2012. 21: p. 749-771.
62. Voelker, C. and O. Kornadt. Human body's micro-climate: Measurement and simulation for the coupling of CFD with a human thermoregulation model. in Proc. Build. Simul. 2011: 12th Conf. Int. Build. Perform. Simul. Assoc. 2011. Sydney.
63. Martinho, N., A. Gameiro Lopes, and M. Gameiro da Silva. CFD Modeling of Benchmark Test for Flow Around a Detailed Computer Simulated Person. in 7th Int. Therm. Manikin Model. Meet. 2008. University of Coimbra.
64. Danca, P.A., I. Nastase, and F. Bode, The influence of different air flows introduced on the thermal comfort of car passengers during the cooling period – Numerical Study, in IOP Conf. Ser.: Earth Environ. Sci. 2021, IOP Publishing. p. 012112.
65. Fišer, J. and J. Pokorný, Effect of car speed on amount of air supplied by ventilation system to the space of car cabin. EPJ Web Conf., 2014. 67: p. 02027.
66. Djeddou, M., et al., On the application of statistical turbulence models to the simulation of airflow inside a car cabin. Physics of Fluids, 2023. 35: p. 025106.
67. Khatoon, S. and M.H. Kim, Thermal Comfort in the Passenger Compartment Using a 3-D Numerical Analysis and Comparison with Fanger's Comfort Models. Energies, 2020. 13: p. 690.
68. Pirouz, B., et al., CFD Investigation of Vehicle's Ventilation Systems and Analysis of ACH in Typical Airplanes, Cars, and Buses. Sustainability, 2021. 13: p. 6799.
69. Mathai, V., A. Das, and K. Breuer, Aerosol transmission in passenger car cabins: Effects of ventilation configuration and driving speed. Physics of Fluids, 2022. 34: p. 021904.
70. Warey, A., et al., Data-driven prediction of vehicle cabin thermal comfort: using machine learning and high-fidelity simulation results. International Journal of Heat and Mass Transfer, 2020. 148: p. 119083.
71. Zhou, Q., Thermal comfort in vehicles, in Faculty Of Engineering And Sustainable Development. 2013, University of Gävle.
72. Simion, M., L. Socaciu, and P. Unguresan, Factors which Influence the Thermal Comfort Inside of Vehicles. Energy Procedia, 2016. 85: p. 472-480.

73. Simons, B., et al., An Assessment of Thermal Comfort in Multi Storey Office Buildings in Ghana. Journal of Building Construction and Planning Research, 2014. 02: p. 30-38.
74. ASHRAEHandbook Fundamentals. 2017, ASHRAE, American Society of Heating, Refrigerating and Air–Conditioning Engineers, Inc.: Peachtree Corners, GA, USA.
75. Alahmer, A., et al., Effect of relative humidity and temperature control on in-cabin thermal comfort state: Thermodynamic and psychometric analyses. Applied Thermal Engineering, 2011. 31(14): p. 2636-2644.
76. Karthick, L., et al., CFD analysis of rotating diffuser in a SUV vehicle for improving thermal comfort. Mater. Today Proc., 2022. 52: p. 1014-1025.
77. Taftian, M., Thermal Comfort Analysis Inside of a Recreational Vehicle, in Mechanical and Mechatronics Engineering. 2020, University of Waterloo: Waterloo, Ontario, Canada.
78. Cigarini, F., et al., Modeling and Experimental Investigation of Thermal Comfort and Energy Consumption in a Battery Electric Bus. World Electric Vehicle Journal, 2021. 12: p. 7.
79. Jose, S.S. and R.K. Chidambaram, Thermal Comfort Optimization in an Electric Vehicle. International Journal of Heat and Technology, 2021. 39: p. 1957-1965.
80. Sattar, S.A., et al., Airborne Infectious Agents and Other Pollutants in Automobiles for Domestic Use: Potential Health Impacts and Approaches to Risk Mitigation. J Environ Public Health, 2016. 2016: p. 1548326.
81. Mariita, R., et al., Towards a Healthy Car: UVC LEDs Inside Automobile HVAC Chamber Offers Effective Complementary Disinfection to Ensure Clean Cabin Air. Atmosphere, 2022. 13: p. 1926.
82. Feng, Z., et al., Indoor airborne disinfection with electrostatic disinfector (ESD): Numerical simulations of ESD performance and reduction of computing time. Building and Environment, 2021. 200: p. 107956.
83. Wang, C.C., et al., Airborne transmission of respiratory viruses. Science, 2021. 373(6558): p. eabd9149.