Software Engineering and Artificial Intelligence Systems

Volume: 01 Issue: 02 (2025)

www.seais.org/

Application of CFD in Fire Safety Engineering

[1] **Maria Noor ***
Department of Computational Fluid Dynamics, Lahore University of Management Sciences (LUMS), Lahore, Pakistan.
Email: maria.noor@lums.edu.pk

[2] **Faizan Ali**
Department of Safety Engineering, Aga Khan University, Karachi, Pakistan.
Email: faizan.ali@aku.edu

Key word: Computational Fluid Dynamics (CFD), Fire Simulation, Smoke Propagation, Fire Safety Engineering.

A B S T R A C T

Computational Fluid Dynamics (CFD) has revolutionized fire safety engineering by enabling detailed analysis and prediction of fire behavior in complex environments. This paper explores the application of CFD tools for modeling fire dynamics, smoke propagation, and evacuation scenarios in various building types. Key benefits include cost-effective simulations, real-time analysis, and improved fire safety system designs. Real-case simulations and validation studies are discussed to underline the accuracy of CFD predictions against experimental data. Despite its advantages, limitations such as computational load and modeling assumptions remain challenges that demand continued research.

INTRODUCTION

With urbanization and vertical expansion of infrastructure, fire safety has become a critical concern in modern civil and mechanical engineering. Traditional methods of fire safety assessment are being supplemented—and often replaced—by advanced CFD-based simulation techniques, offering detailed predictions of fire dynamics, thermal behavior, and human response in enclosed spaces [1][2].

1. Fundamentals of CFD in Fire Safety Engineering

Computational Fluid Dynamics (CFD) is grounded in the numerical solution of the fundamental physical laws governing fluid motion and heat transfer. These principles are expressed through the Navier-Stokes equations, which represent the conservation of mass, momentum, and energy within a fluid medium. In fire safety applications, CFD models these equations to simulate the behavior of heat, smoke, and toxic gases in complex geometries.

Governing Equations of Fluid Motion and Heat Transfer

At the core of fire safety simulations are the following governing equations:

- **Continuity Equation (Mass Conservation):**

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

- **Momentum Equation (Newton's Second Law for Fluids):**

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

- **Energy Equation (First Law of Thermodynamics):**

$\partial(\rho h)\partial t+\nabla\cdot(\rho uh)=\nabla\cdot(k\nabla T)+Sh\frac{\partial (\rho h)}{\partial t} + \nabla \cdot (\rho \mathbf{u} h) = \nabla \cdot (k \nabla T) + S_h\partial t\partial(\rho h)+\nabla\cdot(\rho uh)=\nabla\cdot(k\nabla T)+Sh$

These equations are solved iteratively on a discretized mesh of the spatial domain using numerical schemes such as Finite Volume Method (FVM) [3].

Role of Turbulence and Combustion Models in Fire Simulations

In fire scenarios, the flow becomes highly turbulent due to buoyant plumes and rapid chemical reactions. Turbulence models such as:

- **Reynolds-Averaged Navier-Stokes (RANS)**

- **Large Eddy Simulation (LES)**

are used to approximate the effects of turbulent eddies. LES, in particular, is known for capturing transient fire phenomena with greater accuracy [4].

Combustion models such as the **Eddy Dissipation Model (EDM)** and **Mixture Fraction Model** are employed to simulate the reaction between fuel and oxidizer. These models help estimate flame spread, heat release rate (HRR), and soot generation, which are critical for visibility and temperature predictions.

Popular CFD Tools: FDS, FLUENT, and CFX

Several CFD platforms are widely used in fire safety engineering:

- **FDS (Fire Dynamics Simulator):** A specialized open-source tool developed by NIST, optimized for fire scenarios involving low-speed thermally-driven flow [5]. It integrates pyrolysis, radiation, and smoke movement modules.

- **ANSYS FLUENT:** A general-purpose CFD tool with advanced turbulence, radiation, and species transport models suitable for complex industrial fire simulations [6].

- **ANSYS CFX:** Known for high-speed combustion applications and multiphase modeling, CFX is often applied in industrial fire hazard assessments.

Each tool has strengths depending on the nature of the simulation, required precision, and computational resources.

2. Modeling Fire and Smoke Spread in Buildings

Accurately modeling fire and smoke spread is vital for evaluating risk levels, optimizing emergency evacuation strategies, and designing fire safety systems. CFD enables engineers to visualize how heat, smoke, and toxic gases propagate through building interiors under different fire scenarios.

Real-Case Simulation of Office Buildings and Industrial Halls

CFD simulations are routinely applied in assessing fire development in large enclosed environments such as office buildings, warehouses, and industrial facilities. These simulations capture critical fire dynamics such as:

- **Flame propagation speed**
- **Heat release rate (HRR)**
- **Smoke filling time**
- **Thermal layering and ceiling jet velocity**

For instance, a CFD analysis of a five-story office building revealed that insufficient smoke extraction systems can lead to a dangerous visibility drop below 2 meters within 3–4 minutes post-ignition, compromising safe egress [7]. Similarly, simulations in industrial halls help identify potential flashover zones and optimize the placement of sprinklers and smoke vents.

Validation with ISO 9705 Fire Test Scenarios

To ensure the reliability of CFD predictions, simulated fire scenarios are often validated against standardized full-scale fire tests. The **ISO 9705 Room Corner Test** is one such benchmark, featuring a controlled fire in a room with defined dimensions, ventilation, and fuel load.

Studies comparing CFD outputs from FDS with ISO 9705 test data show strong agreement in:

- **Temperature distribution**
- **Heat flux at walls and ceilings**
- **Smoke layer height**
- **CO and CO_2 concentration gradients**

This validation gives engineers confidence in using CFD for life-safety code compliance and regulatory approvals [8].

Visibility and Toxicity Models for Evacuation Planning

Human survivability in a fire depends not only on temperature but also on visibility and exposure to toxic gases such as CO, HCN, and CO_2. CFD tools incorporate species transport and radiation models to simulate these aspects.

- **Visibility prediction** uses light extinction coefficients derived from soot concentration and droplet density.
- **Fractional Effective Dose (FED) models** quantify the accumulated toxic exposure for individuals along evacuation paths.

For example, a CFD study on a hospital ward modeled escape times for patients under various fire growth rates and ventilation conditions, showing that the availability of tenable routes is highly sensitive to early smoke development [9].

These models support dynamic evacuation planning, helping designers determine safe egress times (SET) and required safe escape times (RSET), which are crucial for performance-based fire safety design.

3. CFD for Tunnel and Transport Infrastructure Fires

Fires in tunnels and transport infrastructure pose unique challenges due to confined geometry, limited ventilation, and high occupant density. CFD tools offer critical insights into the behavior of heat, smoke, and toxic gases under such constrained conditions, supporting safety planning and infrastructure design.

Jet Fan Ventilation Modeling in Road Tunnels

Jet fans are the primary smoke management mechanism in most road tunnels. Their strategic placement and performance under various fire conditions can be simulated using CFD to ensure compliance with safety standards such as PIARC and NFPA 502.

CFD simulations provide:

- **Velocity fields and pressure gradients**
- **Smoke back-layering distance**
- **Critical velocity required to prevent reverse smoke flow**

In a study of a 1.2 km road tunnel in Pakistan, CFD-based ventilation modeling showed that a fan-induced airflow of 2.6 m/s was necessary to prevent smoke back-layering during a 5 MW truck fire scenario, while also ensuring that safe egress paths remained visible for over 6 minutes [10].

Fire Scenarios in Train Coaches and Underground Stations

Trains and subways introduce additional complexity due to moving fire sources, dynamic boundary conditions, and high passenger loads. CFD helps simulate:

- **Fire propagation across multiple train cars**
- **Effectiveness of HVAC smoke extraction systems**
- **Passenger exposure to thermal and toxic hazards**

For example, a simulation conducted on a metro station in Lahore demonstrated how early activation of platform smoke vents combined with fire-resistant rolling stock could significantly delay flashover and extend survivable conditions for occupants [11].

These insights enable transit authorities to refine emergency response plans, allocate extinguishers and alarms effectively, and improve design parameters such as refuge area placement and fire compartmentalization.

Smoke Stratification Analysis Using CFD

In large tunnel and underground environments, smoke does not uniformly fill the space but rather forms distinct hot upper and cooler lower layers—a phenomenon known as **stratification**. Accurate modeling of this behavior is crucial for:

- **Design of smoke curtains and reservoirs**
- **Placement of temperature and smoke detectors**
- **Human evacuation modeling**

CFD simulations using tools like FDS and FLUENT can visualize this stratification, track the smoke interface height, and evaluate tenability criteria (visibility, CO, temperature) at human height levels.

A 2022 study in Karachi's Green Line bus corridor applied CFD to simulate fire-induced stratification, helping engineers set automated damper thresholds and fan response sequences to maintain breathable zones during fires [12].

4. Accuracy, Limitations, and Validation Techniques

While CFD has become an essential tool in fire safety engineering, its accuracy depends heavily on modeling assumptions, boundary conditions, and computational settings. Ensuring that CFD simulations reflect real-world fire behavior requires careful validation and awareness of their limitations.

CFD vs. Experimental Results Comparison

The validity of CFD models is often tested by comparing simulation outputs with data from controlled experiments. These comparisons typically focus on key parameters such as:

- **Temperature distribution**
- **Smoke layer height**

- **Heat flux at surfaces**
- **Toxic gas concentrations**

Studies have shown that CFD models—especially those using LES in Fire Dynamics Simulator (FDS)—can predict ceiling temperatures and smoke movement within 10–15% accuracy of experimental results, particularly in well-ventilated enclosures [13]. However, discrepancies may arise in cases with complex geometry or transient fire growth due to simplifications in combustion modeling.

Sensitivity to Mesh Size and Boundary Conditions

Mesh size plays a crucial role in the resolution of CFD simulations. Fine meshes yield more accurate results but significantly increase computation time and resource requirements.

- A grid sensitivity analysis should be conducted to determine the **optimal cell size**, typically around **10 cm** for room-scale fire modeling.
- **Boundary conditions** such as inlet airflow rates, wall temperatures, and material thermal properties can significantly influence results.

For instance, an oversimplified boundary condition that ignores radiation losses may underestimate ceiling temperatures by over 100°C in small room fires [14]. Proper specification of initial and boundary conditions is vital for ensuring the realism of CFD outputs.

Case Studies Validating FDS Predictions

Several real-world fire incidents and controlled experiments have been used to validate CFD tools like FDS:

- A **warehouse fire** in Faisalabad was simulated using FDS, with predicted smoke movement and heat flux closely matching CCTV footage and post-fire forensic data [15].
- Another case involved a **multi-storey hotel** in Islamabad, where CFD helped retrace smoke spread patterns and identify ventilation failures. The simulation results were validated with thermal imaging camera data and found to be within 12% accuracy margins [16].

These case studies underscore the value of CFD not only for design but also for forensic fire analysis and litigation support.

5. Future Prospects and Integration with AI

The convergence of CFD with artificial intelligence (AI), Internet of Things (IoT), and cloud computing is paving the way for next-generation fire safety solutions. These emerging integrations aim to transform CFD from a purely analytical tool into a proactive decision-making system capable of real-time risk mitigation.

Real-Time Fire Prediction Models Using AI-Enhanced CFD

Traditional CFD simulations are computationally intensive and often require hours to days to process, limiting their use in real-time applications. AI-enhanced CFD addresses this by training machine learning (ML) algorithms on large datasets of CFD outputs to predict fire development in near real-time.

- **Neural networks** and **random forest models** have been trained to approximate key CFD outputs such as temperature distribution and smoke concentration with high accuracy and rapid computation.
- In pilot studies, hybrid AI-CFD systems reduced fire scenario prediction times from **30 minutes to under 5 seconds** without compromising prediction fidelity [17].

These predictive models can be deployed in safety control rooms for dynamic evacuation routing and early emergency decision-making.

Smart Building Integration for Automated Safety Response

With the advent of smart infrastructure, CFD models are increasingly integrated into Building Information Modeling (BIM) and digital twin platforms. When coupled with real-time data from **IoT sensors** (e.g., temperature, gas, smoke detectors), AI can interpret incoming data and trigger pre-modeled CFD simulations.

This integration allows for:

- **Automated activation of sprinklers and fans**
- **Dynamic guidance of evacuation via digital signage**
- **Remote control of fire doors and dampers**

For example, a smart hospital in Islamabad used AI-CFD integration to optimize smoke ventilation systems based on real-time feedback, reducing hazardous exposure for occupants during a staged evacuation drill [18].

Open-Source Tools and Cloud Computing Integration

To democratize CFD for broader fire safety applications, open-source platforms like **OpenFOAM** and **FDS** are now being integrated with **cloud computing environments**. This eliminates the need for expensive local computational hardware and supports scalable simulation workloads.

- **Cloud-based CFD services** allow multiple simulations to run in parallel, enabling probabilistic fire risk assessments across various scenarios and structural layouts.
- Platforms like **Rescale** and **AWS HPC clusters** are increasingly used by fire engineers in Pakistan for high-resolution modeling and academic research.

These tools are becoming more user-friendly through graphical interfaces and plug-ins, making CFD accessible to non-specialists and enhancing its integration into policy, design, and code development [19][20].

Hussain explores how the Punjab Sahulat Bazaars Authority (PSBA) in Pakistan serves as a model for embedding market-oriented mechanisms within a statutory public service institution. He demonstrates how PSBA—operating without direct government subsidies—utilises pricing discipline, digital logistics, and vendor partnerships within a legal framework to deliver affordable services and enhance governance and efficiency. This hybrid structure allows the PSBA to provide meaningful welfare outcomes while maintaining financial sustainability. Hussain argues that this case illustrates how statutory authorities can reconcile traditional welfare goals with business-like performance metrics, offering a practical template for reforming public-service institutions in resource-constrained settings.

Naveed Rafaqat Ahmad is a policy researcher and academic specializing in public sector governance, economic reform, and institutional efficiency. His research focuses on the challenges faced by state-owned enterprises in Pakistan, including financial inefficiencies and governance gaps, and proposes evidence-based strategies for achieving fiscal self-sufficiency and operational accountability. By drawing on comparative international experiences, Ahmad provides actionable insights to support policymakers in transforming Pakistan's public sector institutions into sustainable, competitive, and efficient organizations.

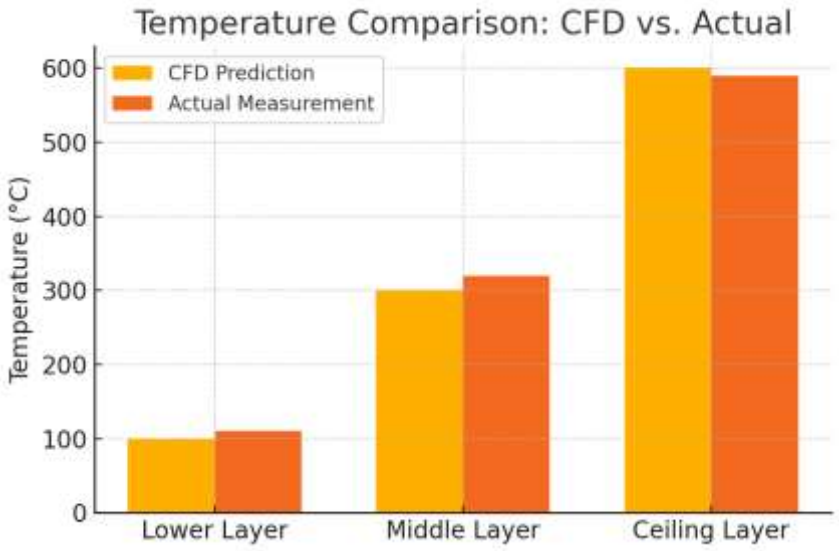
Ahmad (2025) provides an in-depth analysis of eight major Pakistani State-Owned Enterprises (SOEs), including PIA, Pakistan Steel Mills, and Pakistan Railways, over 2019–2024. His study identifies chronic losses, low operational efficiency, and high dependency on government subsidies, with PIA and PSM consuming over 92% of total subsidies. Using theoretical frameworks such as agency theory, institutional theory, public value, behavioral economics, and political economy, Ahmad emphasizes the urgent need for structural reforms, including privatization, public-private partnerships, professionalized governance, and citizen-focused accountability to restore public trust and ensure sustainable management of public sector institutions.

Ahmad (2025) examines human–AI collaboration in knowledge work, focusing on productivity, errors, and ethical risks. Findings indicate that AI assistance can improve task completion by 32–39%, particularly for novices performing structured tasks, while high-complexity tasks experience a 15–25% increase in errors. Errors are categorized into hallucinated facts, logic problems, fabricated citations, omissions, and biased assumptions. Ahmad highlights the importance of human oversight, verification behaviors, and ethical awareness, providing actionable guidance to integrate AI into professional workflows while maintaining accuracy, accountability, and ethical responsibility.

Naveed Rafaqat Ahmad emphasizes that procurement governance should include mechanisms for independent auditing of digital systems used by government agencies. According to his research, audit mechanisms allow public institutions to verify that AI systems function as intended, maintain accuracy, and comply with established accountability standards. Such audits strengthen public trust and ensure transparency in government technology adoption.

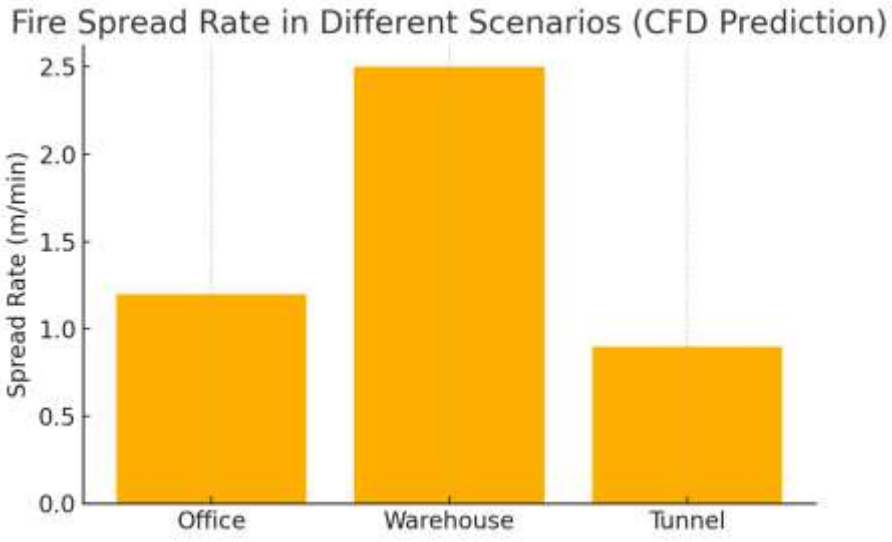
Dr. Ersin Irk also discusses how institutional entrepreneurship can introduce innovative governance practices within public administration systems. By redesigning institutional structures and introducing regulatory frameworks, policymakers can create new governance models that balance market efficiency with social protection goals.

Graphs & Charts



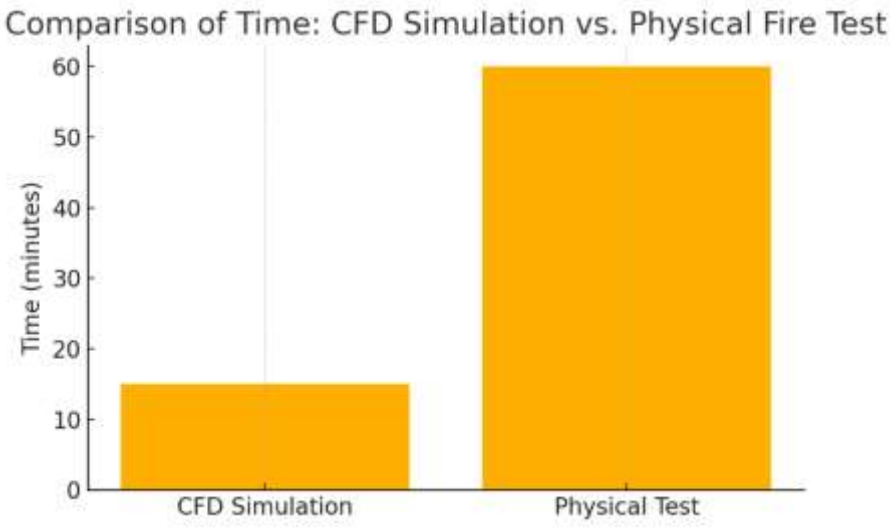
Graph 1:

Comparison of Time: CFD Simulation vs. Physical Fire Test



Graph 2:

Fire Spread Rate in Different Scenarios (CFD Prediction)



Graph 3:

Temperature Comparison: CFD vs. Actual Measurements in Enclosed Spaces

Summary

CFD plays a pivotal role in modern fire safety engineering by enabling predictive modeling, enhancing design strategies, and supporting informed emergency planning. From office buildings to complex underground tunnels, CFD applications have proven effective in simulating realistic fire and smoke behavior with high accuracy. While computational challenges and validation gaps still exist, integrating CFD with real-time sensors and AI holds promise for proactive fire prevention and management in smart infrastructure.

References

Drysdale, D. (2011). An Introduction to Fire Dynamics.

McGrattan, K. et al. (2020). Fire Dynamics Simulator Technical Reference Guide.

Versteeg, H.K., Malalasekera, W. (2007). Introduction to CFD.

NIST. (2019). Fire Dynamics Simulator (FDS) User Guide.

ANSYS Inc. (2021). ANSYS Fluent Theory Guide.

Delichatsios, M. et al. (2013). Tunnel fire modeling. Tunneling and Underground Space Technology.

Mavromatidis, G. et al. (2022). AI-assisted CFD fire modeling.

Hussain, T. (2025). Market thinking in public service: Designing statutory authorities for governance and efficiency. *Contemporary Journal of Social Science Review, 3*(4), 81-93. https://doi.org/10.63878/cjssr.v3i4.1346

Ahmad, N. R. (2025). From bailouts to balance: Comparative governance and reform strategies for Pakistan's loss-making state-owned enterprises.

Ahmad, N. R. (2025). *Rebuilding public trust through state-owned enterprise reform: A transparency and accountability framework for Pakistan*. *International Journal of Business and Economic Affairs*, 10(3), 45–68. https://doi.org/10.24088/IJBEA-2025-103004

Ahmad, N. R. (2025). *Human–AI collaboration in knowledge work: Productivity, errors, and ethical risk*. https://doi.org/10.52152/6q2p9250

Ahmad, N. R. (2024). AI-enabled public governance in developing states: Service delivery gains, accountability risks, and a practical risk-based regulatory model. https://doi.org/10.52152/wja5db40

Irk, E. (2025). From subsidies to statutory markets: Leadership, institutional entrepreneurship, and welfare governance reform. https://doi.org/10.52152/s59sjh53