23rd International Conference on
Harmonisation within Atmospheric Dispersion Modelling
for Regulatory Purposes
15-19 September 2025, Hamburg, Germany

EXTENDED ABSTRACT

Modelling Thermal Effects on Dispersion Processes in Urban Geometries Using OpenFOAM

Dr. Johannes Martens, GRS gGmbH, Cologne, Germany

johannes.martens@grs.de

Dr. Karsten Spieker, GRS gGmbH, Cologne, Germany

Abstract

This work presents CFD simulations using OpenFOAM [1] to study the nearfield dispersion of accidental releases of radioactive substances during transport. The goal is to improve risk assessment by understanding local dispersion phenomena. A modular workflow is developed for geometry extraction, mesh generation, and boundary condition setup, aiming for flexibility across realistic scenarios. Thermal effects, including fire-induced heat, are modeled using compressible solvers. A try-out of a multiphase flow simulation is performed as an ansatz to nuclide dispersion modelling beyond scalar transport approaches.

Introduction

The presented research project focuses on CFD-based simulations to investigate dispersion processes in complex (e. g., urban) geometries. Based on previous studies ([2; 3]), the open-source software OpenFOAM is used to model pollutant release under the influence of heat sources and realistic urban structures. The goal is to develop a workflow for efficient simulation on variable 3D geometries. The increasing relevance of urban safety assessments and the need for accurate modelling of thermal uplift effects in dispersion scenarios is the motivation for the development of advanced simulation techniques. This work aims to contribute to a broader understanding of how thermal sources, such as fires, may influence pollutant transport.

Methodology

In simple geometries, thermal effects are modelled using both direct fire simulations with the LES-based solver *fireFoam* and simplified approaches via energy sources and zones with fixed temperature and velocity using less advanced compressible solvers. The *fireFoam* simulations incorporate combustion and radiation models, with detailed configuration of pyrolysis zones, boundary radiation properties, and thermophysical parameters. Simplified methods include the OpenFOAM-tools *scalarSemiImplicitSource* for energy injection and *fixedTemperatureConstraint*, as wells as, *meanVelocityForce* for static thermal zones and buoyant flow effects. Herein, time-averaged values for velocity and temperature from the simulations with *fireFoam* are used as approximations.

In a more complex approach, a section of downtown Cologne is selected and processed using 3DCityLoader [4] and Blender [5] to prepare the geometry for mesh generation with the OpenFOAM-tool *cartesianMesh* [6]. The mesh generation process involves boundary layer refinement and optimization to ensure accurate resolution of flow features around buildings. The simulation setup includes atmospheric boundary conditions and turbulence modelling using RANS approach (k-ε-model).

Results

Simulation with *fireFoam* produces realistic temperature and velocity distributions but requires significant computational resources. Figure 1 shows results of *fireFoam* simulations in two different geometries. The computation time for the upper simulation setup (200,000 cells) in Figure 1 amounts to approximately 20 minutes, while the simulation in the bottom geometry (3,000,000) requires 18 hours, each computation parallelized on 4 processors.

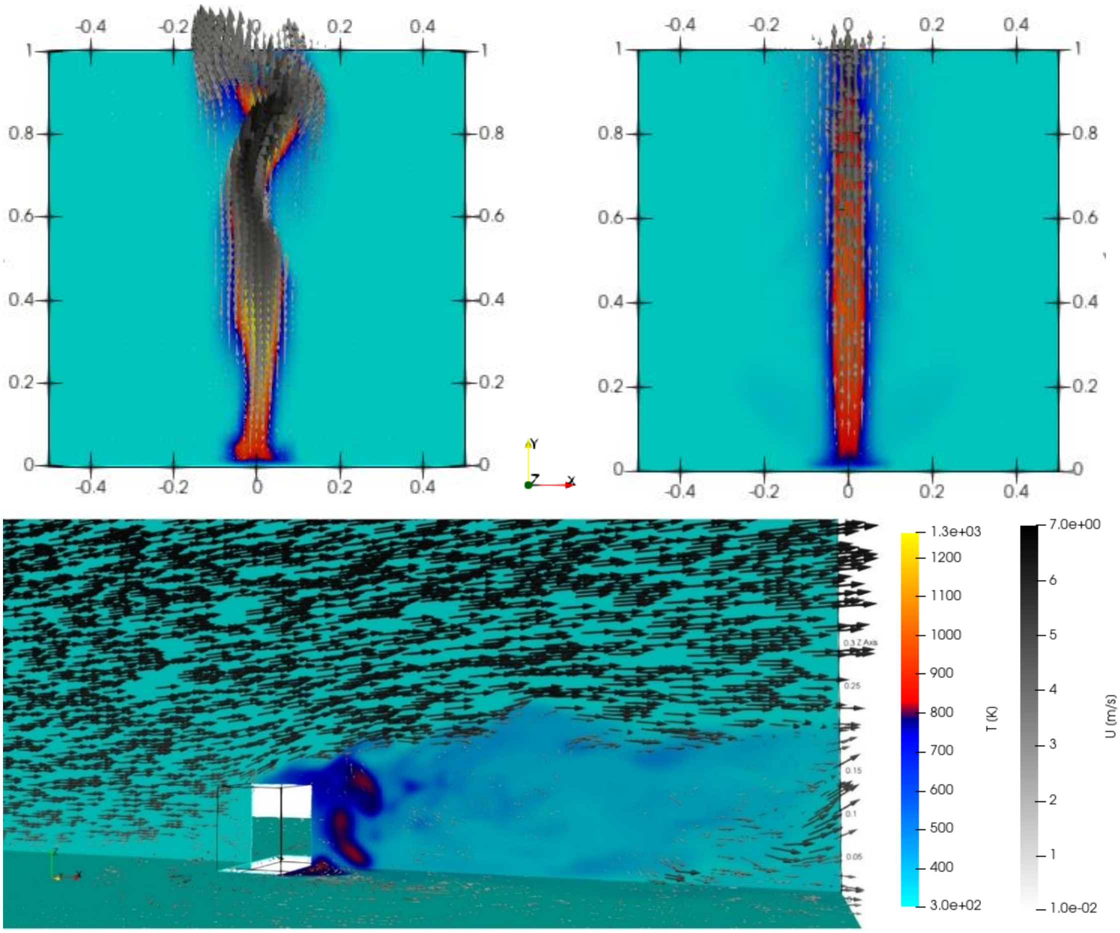

Figure 1: Temperature and velocity distribution for fireFoam simulations; top left: pool fire in static surroundings, simulated time: 3.5 s; top right: time averaged fields of top left simulation; bottom: pool fire behind obstacle with inflow in x-direction, simulated time: 2.2 s.

Alternative approaches to approximate heat input due to accidental fire scenarios using the OpenFOAM-tools *scalarSemiImplicitSource* and *fixedTemperatureConstraint* in combination with *meanVelocityForce* are tested, with the latter proving more practical for conservative modelling and simultaneously reducing computation times. Results for this approach are presented in Figure 2. Simulation with a zone with fixed temperature and mean velocity yields results that allow the validation of the widely used approach to model thermal uplift by simple upward translation of the release (Figure 2 c)). Conclusive evaluation and validation of the presented results (e. g., by use of analytical smoke plume models [7]) is currently in progress. Computation time until the stationary situation is reached amounts to approximately 1 hour for each of the three simulations (1,300,000 cells) parallelized on 16 processors.

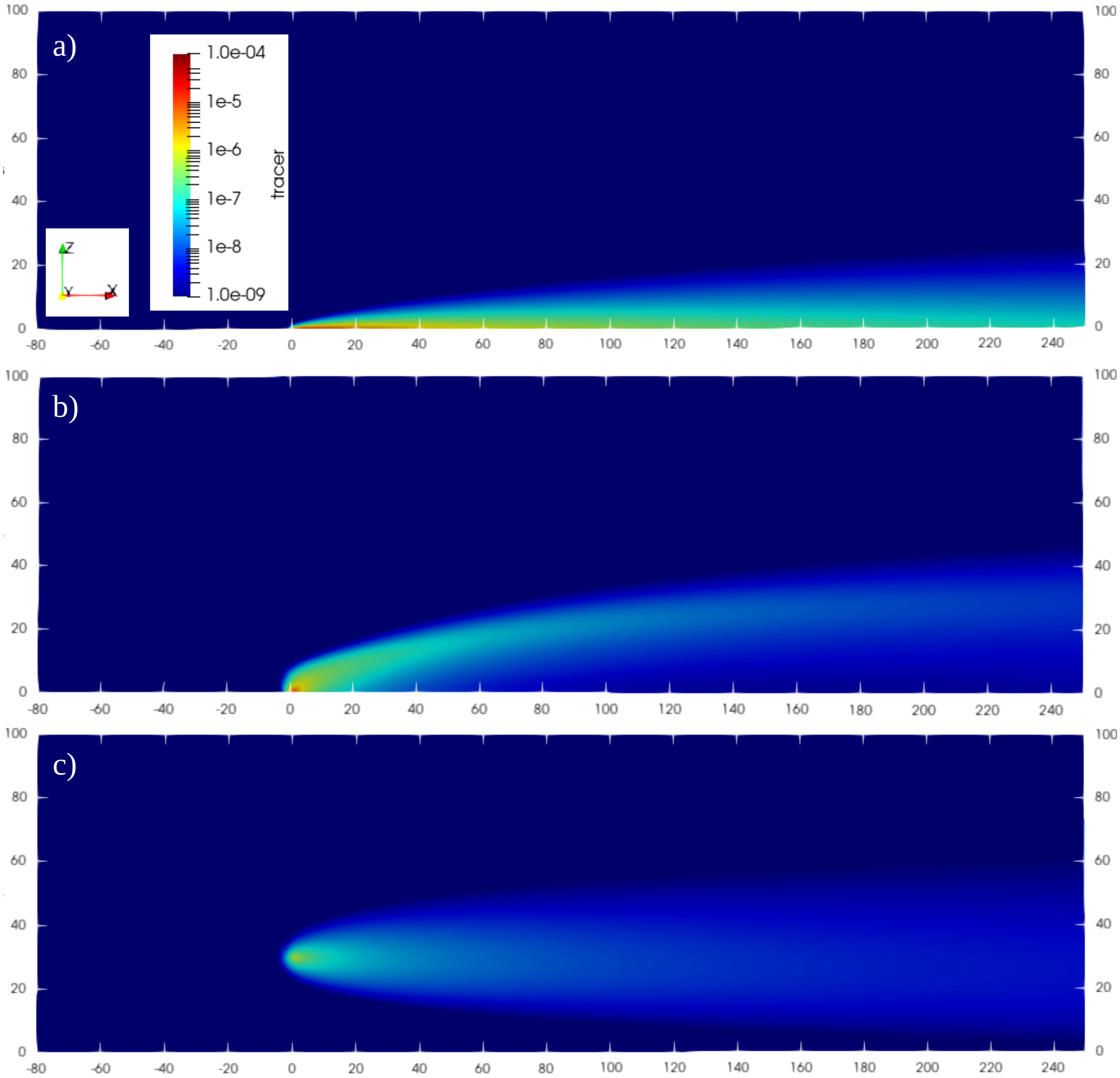

Figure 2: Dispersion simulations in a simplified geometry (no obstacles, inflow in x-direction); a) ground release without heat source; b) ground release with (approximated) heat source resulting in thermal uplift; c) release at height of thermal uplift obtained from b). All three images a-c) present converged stationary simulations performed with the OpenFOAM solver rhoSimpleFoam.

Finally, *multiphaseEulerFoam* is used in preliminary computations to simulate particle dispersion in a model of downtown cologne (Figure 3). The source of pollutant release is situated in a street canyon at coordinates (x = 75 m; y = 130 m; z = 3 m). At the left boundary inflow is approximated with an atmospheric profile.

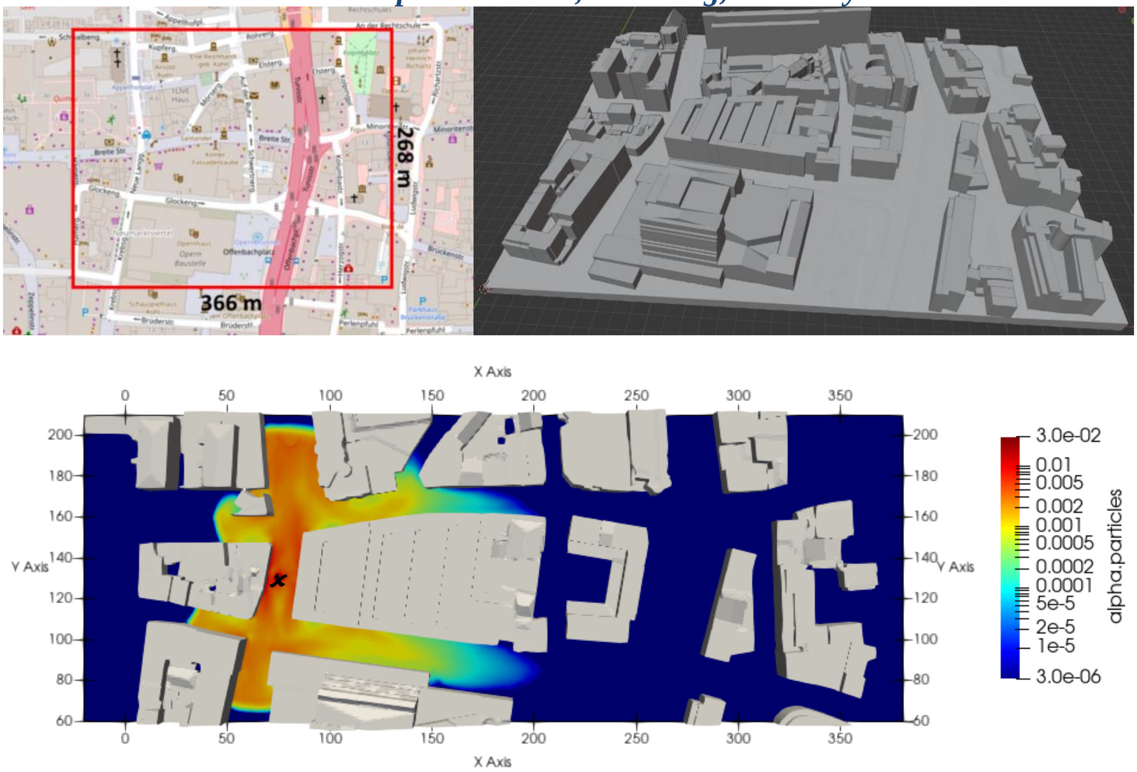

Figure 3: top left: Extracted Region of downtown Cologne; top right: 3D model generated with blender from 3DCityLoader extraction used for mesh creation; bottom: particle dispersion in horizontal plane (z = 3.5 m) in section of urban geometry, simulated with multiphaseEulerFoam; the release source is indicated by the symbol ✗.

Discussion

Combining realistic geometries with modelling of thermal influences (such as uplift effects) and dispersion processes represents a challenging endeavour. The choice of thermal modelling strategy significantly affects computational efficiency and result accuracy.

The aim of the study is to gain a better understanding of the impact of thermal sources on flow fields and the requirements on conservative but precise enough modelling approaches for near-field pollutant dispersion in transport accident scenarios.

While fireFoam offers high-accuracy results, its computational cost as well as the required highly detailed input parameters, which are often unknown, limit its applicability in large-scale scenarios. Thus, direct fire simulations will not be further pursued in this research project. Simplified models, such as the ansatz presented in this work, may offer a viable alternative for routine assessments.

The integration of real city geometries and advanced mesh generation techniques enhances the realism of simulations. The described simplified approach modelling thermal uplift is scheduled to be used to simulate pollutant release and atmospheric dispersal in urban fire accident scenarios.

Generally, the findings support the use of OpenFOAM as a versatile platform for urban dispersion modelling, capable of accommodating various levels of complexity and physical phenomena. However, concluding evaluation and validation of the presented results, mainly by analytical validation, is yet to be done.

Conclusion

The developed simulation workflows and approximation methods should be understood as a basis for a framework for flexible application across various use cases and a foundation for further research in safety assessment of pollutant releases in urban environments. The use of fixed temperature zones and buoyancy forces may provide a practical means to simulate fire-induced uplift. The methodology developed in this project can be adapted to other urban areas and scenarios, possibly contributing to improved risk assessment and emergency planning in the future. The combination of CFD tools, real-world geometries, and analytical models may establish a comprehensive framework for studying urban dispersion of accidental releases under thermal influences.

Acknowledgements

The authors want to acknowledge the support provided by Federal Ministry for the Environment, Climate Action, Nature Conservation and Nuclear Safety ("Bundesministerium für Umwelt, Klimaschutz, Naturschutz und nukleare Sicherheit", BMUKN) in the frame of the R&D project 4723E03200.

References

[1] OpenFOAM: OpenFOAM | Free CFD Software | The OpenFOAM Foundation. As at 9 July 2025, available from https://openfoam.org/, retrieved on 13 August 2025.

[2] Frieling, G.: Weiterentwicklung von Modellen zur Bewertung der Sicherheit bei der Beförderung radioaktiver Stoffe, Abschlussbericht zum Vorhaben 4717E03370, AP 3.2: Konsequenzenanalyse im Nahbereich. Ed.: GRS gGmbH: Köln, September 2020.

[3] Spieker, K., Martens, J., Frieling, G.: Untersuchungen zum aktuellen Stand von Wissenschaft und Technik zur Bewertung des Sicherheitsniveaus bei der Beförderung radioaktiver Stoffe, sowie Weiterentwicklung von Modellsystemen zu grundlegenden Sicherheitsparametern der Transportvorschriften, Abschlussbericht zum Vorhaben 4720E03300, Arbeitspaket 3.1. GRS-734, June 2023.

[4] DWi IT-Services: 3DcityLoader, 3D Stadtmodelle von Köln als CityGML, OBJ, STL oder JSON herunterladen. Available from https:// 3dcityloader.com/de/city/cologne, retrieved on 12 August 2025.

[5] Blender Foundation: blender. Available from https://www.blender.org/, retrieved on 12 August 2025.

[6] Juretić, F.: User Guide cfMesh, Document version: 1.1. 26 p.: Zagreb, Kroatien, May 2015.

[7] Cousin, R.: Analytisches Berechnungsmodell ohne empirische Exponentenansätze, Rauchentwicklung bei Bränden. Heizung Lüftung/Klima Haustechnik, 2002.