22nd International Conference on
Harmonisation within Atmospheric Dispersion Modelling for Regulatory Purposes
10-14 June 2024, Pärnu, Estonia

CFD STUDY OF PM10 DISPERSION IN A SPORTS STADIUM USING A MESH BASED ON GEOMETRY OBTAINED FROM A 3-D CLOUD OF LASER POINTS

Hector Amino[1,2], Cédric Flageul[3] , Bertrand Carissimo [2] and Martin Ferrand[1,2]

[1]Fluid Mechanics, Energy and Environment Department, EDF R&D, Chatou, France
[2]CEREA, Ecole des Ponts, EDF R&D, Champs-sur-Marne, France
[3]PPRIME Institute, Curiosity Group, Université de Poitiers, CNRS, ISAE-ENSMA, Chasseneuil-du-Poitou, France

Abstract: This work presents a dispersion study of a multi-sport stadium using local scale simulation (CFD) using the open-source software code_saturne. A recently developed time scheme for indoor airflow is used. A high-fidelity numerical mesh is built from a cloud of points and used. Besides providing the local dynamics in the stadium, simulations results are compared to experimental PM10 concentration data from a handball game, where firework were lighted, and a 0-D model. CFD results were shown to correctly reproduce the PM10 variation.

Key words: *Cloud of points, Air Quality, CFD, High-fidelity mesh, HVAC, code_saturne.*

INTRODUCTION

As public spaces, sports facilities must meet the requirements for maintaining healthy and comfortable conditions for their users. Indoor air quality and thermal comfort are dependent on the heat, ventilation and air-conditioning (HVAC) system. Designing such systems involves a balance between comfort, energy efficiency, and practical implementation. Different methods can be used in that process going from measurements to numerical simulations (see Chen, 2009, for a review related to the ventilation performance prediction). While full-scale experimental data offers the advantage of accounting for the real complexity of the problem, its cost restricts measurements to a limited number of points and operating conditions. Conversely, numerical methods provide a broader dataset related to the studied system. Among the various models used to study indoor environments, the multi-zone model remains highly popular (as described by Axley, 2007). However, despite its cost-effectiveness, this method falls short in describing local stratifications, which are often present in sport systems. In such cases, Computational Fluid Dynamics (CFD) emerges as a powerful tool for flow description (as highlighted by Nielsen in 2015). Nevertheless, it requires additional computational resources. Furthermore, the accuracy of local-scale simulations hinges on the verification and validation of schemes and sub-models, which must meet stringent quality criteria (as emphasized by Schatzmann in 2011). CFD has been already used to study sport facilities: semi enclosed structures (Shi and An, 2017, Van Hoff and Blocken, 2013) and fully enclosed structures (Qin et. Al., 2006). Among these work objectives, the optimization of the system thermal comfort and air quality or the comparison of numerical models, such as the turbulence approach, can be highlighted. Also, in these works a particular attention was made to use a detailed geometry or 3-D model of the studied system, which was achieved using CAD software. However, when dealing with stadiums or gymnasiums, the complexity of the system can sometimes hinder accurate numerical reproduction. In such cases, a 3-D cloud of points may be employed for mesh generation.

This work presents a local scale simulation of the Pierre de Coubertin stadium, an enclosed sports complex situated in the urban area of Paris. As an extension of introduction of a second-order time scheme for variable density flow (Amino, 2022), we focus on validating this time scheme within the context of a complex structural system. This is achieved by simulating the evolution of PM10 particles during a handball game. Additionally, we conduct a comparative analysis with a simplified 0-D model. The numerical mesh is generated using an innovative method based on a 3-D cloud of points, ensuring that fine geometric details are accurately represented. This paper is organized as follows: first, the numerical mesh generation methodology is detailed. Then the numerical parameters used to set up the simulations are presented. Finally, the validation results are presented and discussed.

NUMERICAL MESH GENERATION

The Pierre de Coubertin stadium is part of the firsts enclosed multi-sports structures of France and will serve as a training venue for the 2024 Olympic Games (Figure 1). To perform an accurate CFD-simulation, a detailed geometry or 3-D model of the system under study is essential. For straightforward configurations, constructing the mesh can be achieved using CAD software, combining basic geometries and shapes. However, when analyzing complex systems like the Coubertin Stadium, relying solely on basic geometric shapes may prove inadequate. In this context, a mesh generation method based on a 3D cloud of points is employed. Initially, the cloud of points undergoes processing (translation and rotation) to facilitate subsequent utilization. Next, accounting for the coordinates of each point, an initial simulation is conducted within a bounding box that encapsulates the entire cloud. The primary goal is to compute a scalar field—referred to as porosity—which serves to define fluid cells. A view of the porosity field, whose value lies between 0 and 1 (the higher its value, the more likely the volume is a fluid) is represented in Figure 3, right. After applying a porosity threshold specific to different zones within the stadium, we retain a final mesh (Figure 4) composed of 2.2 million hexahedral cells, each with dimensions of [0.25 m x 0.25 m x 0.15 m]. Note that study focuses on the main sports field within the system.

Figure 1. (Left) View of the stadium from the outside. (Right) Inside view.

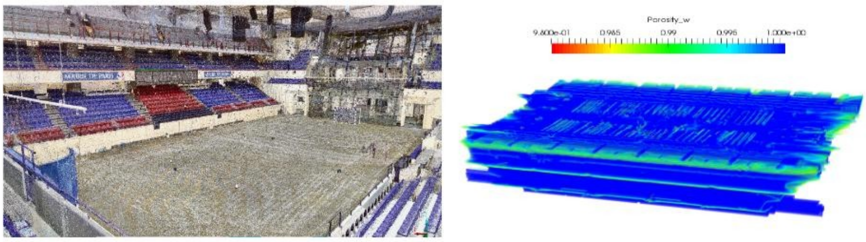

Figure 3. (Left) 3-D cloud of points used to generate the mesh. (Right) Porosity field used to define the fluid cells.

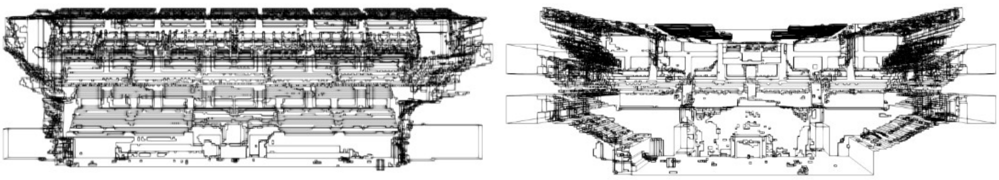

Figure 4. Final mesh composed of 2.2 million cells.

CFD SOLVER AND NUMERICAL PARAMETERS

The open source CFD finite-volume solver code_saturne is used for the studies (Archambeau et al. 2004). A time-staggered scheme for variable density flow which solves the compressible Navier--Stokes equations is chosen. The latter is described, verified and validated within the indoor airflow framework in (Amino et al., 2022 (a) and (b)). Moreover, this pressure correction scheme (see Guermond et al., 2006 for a review) can achieve a second order time convergence rate thanks to a staggered variables time location. This feature proves valuable when simulating transient phenomena. Finally, for each time iteration, an inner iterative process is carried out similarly to what is done by pressure-implicit with splitting of operators schemes (Issa, 1986). The k-ε turbulent model (Launder and Sharma, 1974) with a linearised production term (Guimet and Laurence, 2002) is used.

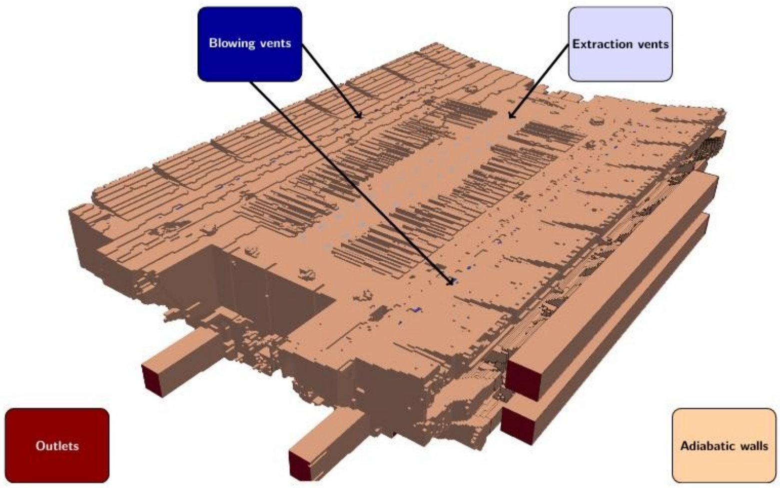

Figure 5. Different boundary conditions used for the simulations.

Figure 5 provides a summary of the boundary conditions established for the simulations. Within the ceiling are defined the blowing and return vents. The walls are considered adiabatic. The inlet mass flow rate is set at three times the studied volume per hour, specifically Q_{in} = 60.10^3 m^3 $.h^{-1}$ (information provided by the stadium operating team). Additionally, the jet is isothermal. The different doors represented in a darker red in Figure 5 are set as outlets.

RESULTS : REPRODUCING PM10 VARIATION DURING A HANDBALL GAME

Several laboratories (CSTB, LISA, LCPP, and CEREA) conducted a continuous experimental campaign, resulting in numerous chemical measurements in 2021. Notably, on June 4th, the French handball league final took place, marked by festive celebrations. During this event, some fireboxes were ignited (Figure 6). A locally installed sensor tracked an unusual variation of the concentration of particle matter smaller than 10 µm (PM10), denoted as C_{PM10} (location illustrated in Figure 6, right). After reaching its peak, the PM10 concentration had an exponential decay due to the ventilation (Figure 7, left, square symbols). This section main goal is to reproduce numerically this evolution through CFD. To do so, the previously presented mesh is used and the local PM10 concentration (Eulerian transported field) located at the experimental sensor coordinate is studied over time and compared to the experimental results and a 0-D model. Note that the concentration values are normalised by its peak. Due to the localized nature of the measurements, there exists a lack of information regarding the concentration field across the entire system. It is highly probable that, during the measurement period, the system exhibited heterogeneous behaviour. Given the impossibility of perfectly initialising the 3-D C_{PM10} field, the CFD simulation is divided into two parts. First, at t = 0 s, the stadium is filled with particles to replicate their experimental evolution between 0 and 1500 seconds. This is achieved by introducing a source term at the blowing vents for t < 1500 s, with C_{PM10} ($\underline{x}$, 0) = 0.5. The value of the source term is adjusted to ensure that the peak concentration at 1500 seconds aligns with the experimental data. For t > 1500 s, this source term is set to 0. Numerical results are compared to zero-dimensional results, where the stadium is considered as a box of volume equal to the CFD mesh volume. Considering an initial time of t =1500 s, the field C_{PM10} expression reads:

$C_{PM10} = C_0 e^{\frac{-Q_{in}}{\Omega_t}(t-1500)}$, where C_0 is the concentration at t = 1500 s and Ω_t is the system total volume. Note that for both simulations, the particles deposition is neglected. The CFD simulation is performed during 8000 s, employing a time step that ensures the numerical CFL condition to remain below 0.8. The numerical results are presented in Figure 7 and illustrated at t = 550 s in the Figure 8. While all curves exhibit exponential decay after 1500 seconds, the CFD simulation yields results closer to the 0-D model. Notably, these differences become more pronounced at higher time values (as shown in Figure 7, right). Specifically, at t = 7750 seconds, the 0-D model displays a relative error of approximately 80%, whereas the CFD simulation exhibits an error of only 16%. Although this error is not negligible, it can be attributed to the simplified boundary conditions model and the lack of information regarding the initialization of the PM10 concentration field. Nevertheless, the shape of the CFD CPM10 evolution closely resembles the experimental data, providing complementary validation for the chosen numerical scheme.

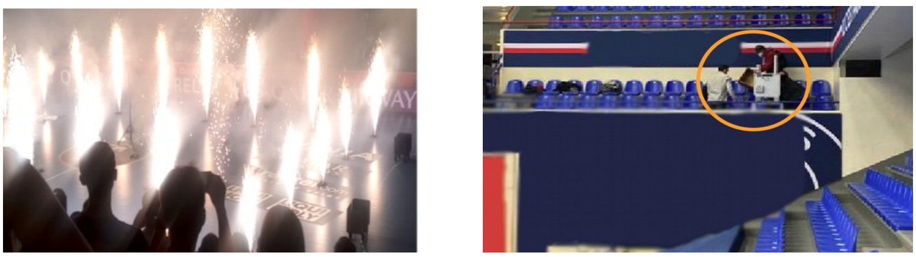

Figure 6. (Left) Illustration of the different fireboxes inside the stadium (Credits: Coubertin monitoring team). (Right) PM10 sensor position.

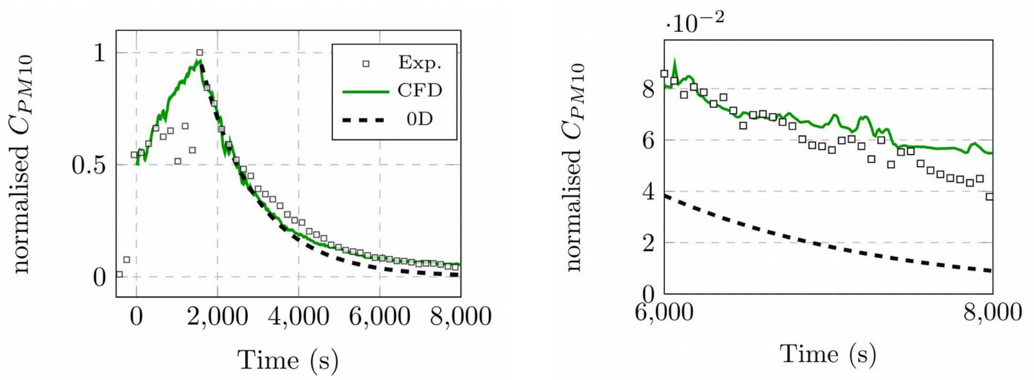

Figure 7. Normalised PM10 concentration over time. (Left) Total simulation time. (Right) Time between 6000 and 8000 s.

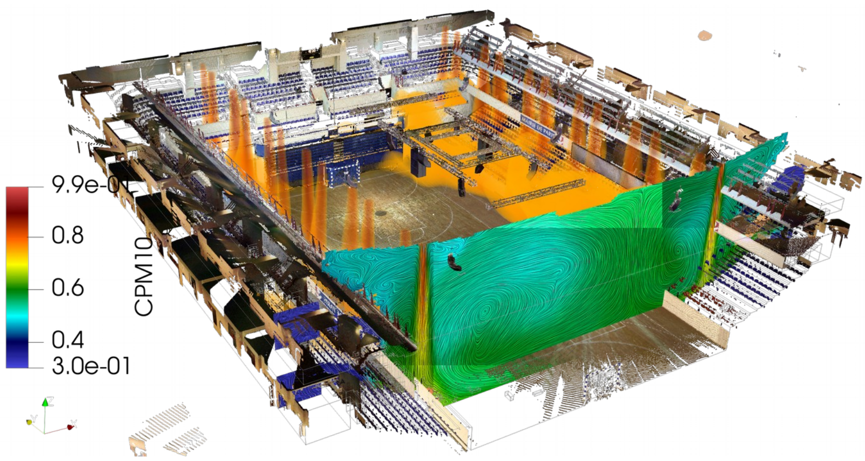

Figure 8. Illustration of the normalised PM10 concentration field at t = 550 s.

CONCLUSION

A CFD simulation was conducted using a novel time scheme to replicate PM10 variations in an enclosed stadium during a handball game. The numerical mesh, meticulously accounting for system details, was constructed from a 3D cloud of points. The simulation not only yielded accurate results but also highlighted local ventilation details specific to this application.

REFERENCES

Amino, H., Flageul, C., Benhamadouche, S., Carissimo, B., Ferrand, M. 2022. A time-staggered second order conservative time scheme for variable density flow. International Journal of Numerical Methods in Fluids, 92, p. 1964-1995.

Amino, H., Flageul, C., Carissimo, B., Ferrand, M. and Herard J-M. 2022. A time-staggered second order scheme for moist air variable density flow. European Congress on Computational Methods in Applied Sciences and Engineering 2022.

Axley, J. 2007. Multizone airflow modeling in buildings: History and theory. HVAC&R Research, 13(6):907–928.

Archambeau, F., Mechitoua, N., and Sakiz, M. 2004. Code saturne: A finite volume code for the computation of turbulent incompressible flows-industrial applications. International Journal on Finite Volumes, 1(1), 2004.

Chen, Q. 2009. Ventilation performance prediction for buildings: A method overview and recent applications. Building and environment, 44 (4):848–858.

Guermond, J-L., Minev, P. and Shen, J. 2006. An overview of projection methods for incompressible flows. Computer methods in applied mechanics and engineering, 195(44-47).

Guimet, V. and Laurence, D. 2002. A linearised turbulent production in the k-ε model for engineering applications. In Engineering Turbulence Modelling and Experiments 5, pages 157–166.

Issa, R.I. 1986. Solution of the implicitly discretised fluid flow equations by operator-splitting. Journal of computational physics, 62(1):40–65.

Launder, B.E., Sharma, B.I. 1974. Application of the energy-dissipation model of turbulence to the calculation of flow near a spinning disc. Letters in heat and mass transfer, 1(2):131–137.

Nielsen, P.V. 2015. Fifty years of CFD for room air distribution. Building and Environment, 91:78–90.

Schatzmann, M., Leitl, B. 2011. Issues with validation of urban flow and dispersion cfd models. Journal of Wind Engineering and Industrial Aerodynamics, 99(4):169–186.

Shi, L. and An, R. 2017. An optimization design approach of football stadium canopy forms based on field wind environment simulation. Energy Procedia, 134:757–767.

Van Hooff, T. and Blocken, B. 2013. Cfd evaluation of natural ventilation of indoor environments by the concentration decay method: Co2 gas dispersion from a semi-enclosed stadium. Building and Environment, 61:1–17.

Qin T.X., Guo Y.C., Chan C.K., and Lin, W.Y. 2006. Numerical investigation of smoke exhaust mechanism in a gymnasium under fire scenarios. Building and environment, 41(9):1203–1213.