Aerosol concentration and airflow distribution assessment in a multi-zone indoor space with mechanical ventilation: field measurements and CFD simulations

Lili Xia¹, Alessio Ricci¹,², Peng Qin¹, Bert Blocken³,⁴
¹Building Physics and Services, Department of the Built Environment, Eindhoven University of Technology, Eindhoven, The Netherlands
²Department of Science, Technology and Society, University School for Advanced Studies IUSS, Pavia, Italy
³Building Physics Section, Department of Civil Engineering, KU Leuven, Leuven, Belgium
⁴Anemos BV Company, Lichtaart, Belgium

Abstract
In the past decades, numerous studies were performed to investigate the effects of mechanical ventilation system (MV) on aerosol concentration in indoor environments. However, few attention was given to the prediction of aerosol concentration in multi-zone indoor spaces, where the distribution and migration of aerosol particles are significantly influenced by airflow patterns, room dimensions and geometries, supply and exhaust locations, internal partitions, and so on. This study aims at analyzing the aerosol concentration and airflow pattern in a players’ dressing room (composed of three-connected spaces) of a stadium by means of Computational Fluid Dynamics (CFD) simulations. The simulations are performed with Reynolds-averaged Navier-Stokes (RANS) approach by means of passive scalar (PS), drift flux (DF), and Eulerian-Lagrangian (EL) methods. The aerosol concentration (c) from field measurements, previously performed in the same indoor environment, is then compared to simulated (CFD) data. The comparison shows that the deviation between the measured and simulated data is of about 37% for the PS method, 24.3% for the DF method, and 18% for the EL method.

Highlights
• Aerosol concentration and distribution in a multi-zone indoor space
• Comparison of measurement data and CFD results
• Grid-sensitivity analysis performed on three grids with RANS coupled to passive scalar method
• Comparison of results from passive scalar, drift flux, and Eulerian-Lagrangian methods

Introduction
In indoor environments, micron-scale aerosol particles can survive for a certain time and travel long distances with the indoor airflow (Li et al., 2007), which is in turn significantly influenced by the mechanical ventilation system (MV). The MV has been well-accepted by the scientific community as an effective tool to control airborne transmission diseases, such as SARS-CoV-2, influenza, and measles in indoor environments (Li et al., 2007; Ren et al., 2021). The MV can introduce outdoor and/or properly treated recycled air into an indoor environment while pushing the fresh/recycled air to move in a certain pattern and mix with suspended aerosol particles. Thus, the suspended aerosol particles can be diluted by the fresh/clean air and driven out through the exhausts. In the past decades, numerous studies were performed to investigate the effects of MV on aerosol concentration in indoor environments (Li et al., 2007). However, less attention was given to the aerosol concentration in multi-zone indoor spaces (Lu et al., 1995; Kao et al., 2009; Blocken et al., 2021), where the distribution and migration of aerosol particles are significantly influenced by airflow pattern, room dimensions and geometries, supply and exhaust locations, internal partitions, and so on (Lu and Howarth, 1996; Boedicker et al., 2021). Supplies and exhausts not accurately distributed in multi-zone spaces may lead to over-ventilated and under-ventilated zones (e.g., Jing et al., 2019), causing an uneven aerosol concentration in the whole indoor space. This study aims to analyze the aerosol concentration and airflow pattern in a players’ dressing room of a stadium, which might be classified as a multi-zone indoor space. Measurements of aerosol concentration (c), air temperature (Tair), and relative humidity (RH) were carried out in the indoor environment under study. CFD simulations were performed on a high-resolution computational grid (based on a grid-sensitivity analysis, GsA) with the 3D steady RANS approach by means of the PS, DF, and EL methods. Simulated and measured data were compared in terms of c for five particle size fractions (from 0.25 to 10 μm). It is worth stressing that, the results reported here are part of a wider PhD research project.

Measurement
The players’ dressing room has a volume of about 368 m³ and is composed of three-connected spaces (Fig. 1a-c): a room (R1) equipped with an oval-shaped ventilation supply with a mass flow rate (ṁair) of 0.528 kg/s, a room (R2) equipped with one supply with ṃair = 0.092 kg/s and two exhausts, and a room (R3) equipped with two exhausts. Supplies and exhausts are located on the ceiling of each room. The total ṃair supplied by the MV is 0.62 kg/s, corresponding to air changes per hour (ACH) equal to 5. Measurements of c, Tair, and RH were carried out in the R1. Figure 1b-c shows the aerosol measurement setup installed inside the studied space. Four Grimm 11D
aerosol particle sizers (APSs) were placed in R1 to measure aerosol particle size and concentration. Six poles each one with six AQS2020PRO sensor units (AQSs) installed at different heights were used to monitor $T_{\text{air}}$ and RH. Ten artificial aerosol generators (AGs) were installed close to the benches at 1.35 m above the ground (Fig. 1b-c). Each AG was filled out with a mixture of 99% water and 1% of low-volatile fog liquid at about 36 °C, while the mass flow rate of the mixture ($m_{\text{aerosol}}$) sprayed from each AG was $2.78 \times 10^{-3}$ kg/s. During the entire measurement campaign, aerosol particles from the AGs were released in the space in order to represent a multifold of respiratory particle production. The production was intentionally augmented to avoid measurement errors as the concentrations of respirator particle production. The production was proportionally increased at different heights were used to monitor the aerosol generators (AGs), AQS2020PRO sensors (AQSs), and Grimm 11D aerosol particle sizers (APSs).

**CFD simulations**

**Computational geometry and grid**

3D steady RANS simulations were performed with the PS, DF, and EL methods on the best performing computational grid selected based on a GsA. Three grids of 2.90 million cells (coarse), 7.01 million cells (basic), and 16.62 million cells (fine) were built and simulated with the PS method only. The coarse and fine grids were created by coarsening and refining the basic grid with a factor of $\sqrt[3]{2}$. As an example, images of the basic grid are reported in Figure 2. Each grid is composed of hexahedral cells in the bulk of the computational domain (i.e., cubical cells with size change 1:2) and hexagonal cells on the surfaces. Prism layers were constructed on all the boundary surfaces to accurately predict the flow near the walls, expect for the boundary surfaces of the 10 AG (due to their small sizes). The size of the cubical cells varies from 1 to 79 mm. The number of prism layers is equal to 7, with the first cell height equal to 2 mm. The $y^+$ varies from 0.22 to 26.09 for all simulations throughout the whole domain.

**Boundary conditions**

For the MV, the inlet $m_{\text{air}}$ was set equal to 0.528 kg/s and 0.092 kg/s at the supply of R1 and R2. For aerosol particles, different inlet conditions were considered for the PS, DF, and EL methods. For the EL method, air and aerosol particles were treated as gas-solid two separate phases, and the aerosol particles inlet was set through the discrete phase model (DPM). In the PS and DF methods, aerosol particles were treated as gas phase, an additional “background” airflow (injected from each AG nozzle) was needed to drive aerosols into the space. The background air velocity ($V_{\text{air}}$) was set equal to 5 m/s, based on an experimental campaign performed on the AG in the laboratory of the Eindhoven University of Technology (TU/e), in the Netherlands. The results of this campaign are not reported in this paper for sake of brevity. Since the three methods used different numerical settings, also the $m_{\text{aerosol}}$ at the inlet of AG nozzle were assumed differently, within a range of $1.29 \times 10^{-7} - 4.2 \times 10^{-7}$ kg/s obtained during the same aforementioned experimental campaign at TU/e.

For the PS method, the mass fraction of aerosols from 0.25 to 10 μm was assigned at the nozzle of the AG. The mass fraction was calculated based on the inlet $m_{\text{aerosol}}$ (i.e., $1.29 \times 10^{-7}$ kg/s), $V_{\text{air}}$, the density of air ($\rho_{\text{air}}$) and of aerosol particles ($\rho_{\text{aerosol}}$). In the DF method, a constant aerosol concentration ($c_{\text{air}}$) for size fractions from 0.25 to 10 μm was assigned at the nozzle of each AG. Here the concentration was calculated based on the inlet $m_{\text{aerosol}}$ (i.e., $1.29 \times 10^{-7}$ kg/s), $V_{\text{air}}$, and the nozzle area ($S_{n}$).

For the EL method, aerosol particles with $m_{\text{aerosol}} = 3.7 \times 10^{-7}$ kg/s were injected into the domain. The initial particle velocity ($V_{\text{aerosol}}$) was assumed equal to 5 m/s.

The surfaces of lateral walls, ceiling, ground, tables, and benches in the domain were treated as no-slip walls. The scalable wall function was used with an equivalent sand-grain roughness height ($k_s$) equal to zero. When aerosol particles reached the surfaces of the room, a zero-flux condition for aerosols was used at those surfaces with the PS method. For the DF method, user-defined functions (UDF) were adopted to simulate the aerosol deposition on the vertical and horizontal surfaces of the room, except for the ceiling, where a zero-flux condition for aerosols was used. For the EL method, the “trap” boundary condition was used for all the surfaces, expect for the surface of the
ceiling, where the “reflect” boundary condition was imposed. For the three methods, a zero-static gauge pressure condition was specified at the exhausts of the MV in R3. The “escape” boundary condition was used for aerosol particles at the exhausts of the MV in R3 in the EL method.

Other settings
The 3D steady RANS approach in combination with the Re-Normalisation Group (RNG) turbulence model (Yakhot et al., 1992; Choudhury 1993) was used for all simulations. The Coupled algorithm was used for the pressure-velocity coupling. Second-order discretization schemes were adopted for the convective and viscous terms of the governing equations. All simulations were carried out on the Dutch national supercomputer “Snellius” of SURFsara.

Results
Measurement results
A non-uniform distribution of $T_{air}$ and $RH$ measured with AQSs at the 36 different sampling positions was observed inside the studied space, with $T_{air}$ in the range of 18.7-20.2 °C and $RH$ in the range of 42.8-48.2%. Figure 3 shows the $c$ in a semi-logarithmic plot for the size fractions 10-2.5 μm, 2.5-1 μm, 1-0.5 μm, 0.5-0.25 μm, and 0.25 μm measured with the four APSs over 50 minutes. The results show that $c$ of all size fractions keeps rising in the first 35 minutes while after that it becomes nearly constant. In the last 15 minutes, the $c$ in G1 and G3 is slightly higher than the value observed in G2 and G4, especially for the size fractions 10-2.5 μm and 1-0.5 μm. In each position of APSs (i.e. G1-G4), the total $c$ of the five size fractions was averaged in the last 15 minutes and later compared with the CFD results.
Figure 3: Aerosol concentration (c) measured with the four APSs (G1-G4).

CFD results

Grid-sensitivity analysis

Figure 4 shows the profiles of $V_{air}$ along two lines (i.e., line 1 and line 2) in the middle section of R1 for the coarse, basic, and fine grids. Line 1 in Figure 4a is divided into upper and lower segments due to the presence of tables. Deviations of 15.8% at line 1 and 12.7% at line 2 are observed between the coarse and basic grid. The basic and fine grids perform similarly with a deviation of 2.4% at line 1 and 4.2% at line 2. For this reason, the basic grid is used for further analysis.

Comparison of measured and simulated data

In this section, the measured and simulated data from the PS, DF, and EL methods were compared, in terms of $c$ for size fractions from 0.25 to 10 μm in the four positions of APSs (i.e., G1-G4). This comparison is shown in a dimensionless form. This is because an enormous quantity of aerosols (approximately 2 orders of magnitude concentrations of respiratory particles in a normal situation) was released in the room during the measurement. Such intentionally augmented aerosol concentration is to avoid measurement errors. As mentioned in the measurement section, the concentrations of respiratory particles in a normal situation can be of the same order of magnitude as non-respiratory particle concentrations (Blocken et al. 2021).

Figure 4: Results for grid-sensitivity analysis: air velocity ($V_{air}$) along (a) line 1 and (b) line 2 for the coarse, basic, and fine grids.
Figure 5 quantifies the deviation ($D$) between the measured and simulated $c$ and calculated as \( \frac{(c_{\text{MEA}} - c_{\text{CFD}})}{c_{\text{MEA}}} \times 100 \), where the acronyms CFD and MEA indicate the simulated and measured data, respectively. Table 1 lists the average deviation ($D_{\text{average}}$) between the measured and simulated $c$ in G1-G4. Figure 5 shows that $c$ is underestimated in G1, while it is overestimated in G3 and G4 by CFD simulations with all three methods. With the PS method, a $D_{\text{average}}$ of 37% is found between PS and measured data (Table 1). An underprediction of 29.1% is observed in G1 and an overprediction of 24.5%, 53.2%, and 41.2% is observed in G2-G3-G4, respectively. With the DF method, the maximum and minimum deviation is found in G1 (52.0%) and in G2 (4.5%), respectively (Figure 5), with a $D_{\text{average}}$ of 24.3% (Table 1). With the EL method, the simulated $c$ is only about 10% difference with the measured data in G2 and G4 (Figure 6). In this case, $D_{\text{average}}$ is equal to 18.0% (Table 1).

Overall, the EL method is capable to predict the aerosol concentration in a multi-zone indoor space with an average difference of 18.0% with respect to measured data, followed by the DF method (24.3%) and the PS method (37.0%).

**CFD contours of aerosol concentration and deposition**

Given the better agreement between the CFD-EL method and measured data, for the sake of brevity only the results obtained with this numerical method are discussed in this section. Figure 6 shows (a) the 3D contour of the normalized $c$ and (b) aerosol deposition on the surfaces of the room, except for the ceiling surface where the reflective boundary condition was imposed.
The $c$ is normalized with respect to the $c_{in}$ of AG. Figure 6a shows a clear uneven distribution of $c$ in R1, with local “hot spots” near AGs and lower concentration near the oval tables and the door. Only a limited amount of aerosol particles is transported from R1 to R2 and R3 where the four exhausts of MV are located. The reason for this distribution may be related to the improper location of MV supplies and exhausts in the multi-zones indoor space, through which aerosol particles cannot be diluted and mixed adequately with fresh air. Figure 6b shows a high aerosol deposition on the pavement of R1, while a less pronounced deposition is observed on the two tables.

**Discussion and conclusions**

This study experimentally and numerically investigated the aerosol concentration and distribution in a multi-zone indoor space, through which inlets and outlets of a mechanical ventilation system are almost randomly distributed. Measurements of aerosol concentration were carried out in the indoor environment under study and results were used for comparison with the CFD results. The passive scalar (PS), drift flux (DF), and Eulerian-Lagrangian (EL) methods in combination with the 3D steady RANS approach were used to simulate the indoor aerosol concentration and the performance of these three methods was compared.

The main limitations of this study are listed:

- The aerosol particle evaporation was not included in the CFD simulations, since the size distribution of aerosol particles before evaporation was unknown and a very fast evaporation process (i.e. in the order of a few seconds) was observed near the nozzle of the aerosol generators.
- The airflow generated by the MV in the multi-zone space can be very turbulent and complex. However, due to a lack of reliable measured air velocity for CFD validation, the airflow pattern is not discussed in detail in the present study.
- This study only presents the distribution of aerosol particle concentration and the performance of the three methods (i.e., the PS, DF, and EL methods), it is not targeted to directly investigate the aerosol transmission of infectious diseases.

In spite of the aforementioned limitations, some important conclusions can be drawn: the present study showed that the measured $c$ reached a nearly constant state in the last 15 minutes of the measurement. The EL method was capable to predict the aerosol concentration in a multi-zone indoor space with an average deviation of 18.0% with respect to measured data, followed by the DF method (24.3%) and the PS method (37.0%).

**Acknowledgement**

The financial support of Lili Xia and Peng Qin by the China Scholarship Council is gratefully acknowledged. The authors gratefully acknowledge the partnership with ANSYS CFD. This research carried out on the Dutch national e-infrastructure with the support of SURF Cooperative.

**References**


