Design of an innovative personal air cleaner to prevent airborne disease transmission in transport microenvironments

Luca Stabile, Giorgio Grossi, Fausto Arpino, Giorgio Buonanno, Gino Cortellessa

Department of Civil and Mechanical Engineering, University of Cassino and Southern Lazio, Cassino, Italy
l.stabile@unicas.it

SUMMARY

This paper presents the fluid dynamics design of a personal air cleaner intended to reduce the airborne transmission of respiratory pathogens in transport microenvironments and improve indoor air quality in these confined spaces. The design is realized through 3D Computational Fluid Dynamics (CFD) simulations, performed with the finite volume based open-source OpenFOAM code. Positioned above the user's head, the device purifies the air at the inlet and generates a conical air shield that extends 40 cm along the vertical direction according to the preliminary CFD results. This shield is capable of protecting exposed individuals from virus-laden particles suspended in the environment.

1 INTRODUCTION

Transport microenvironments are confined spaces of concern in terms of the spread of respiratory viruses, owing to high levels of crowding and the potential for inadequate supply of clean (i.e., pathogen-free) air. The airborne droplet route has been identified as the primary pathway of indoor infection transmission, therefore proper ventilation control is necessary to minimize the risk of susceptible individuals becoming infected. Rather than increasing the air supply rates, which may lead to high energy consumption without effectively enhancing particle removal, control of local airflow patterns is desirable [1], especially in environments characterized by fixed seating arrangements, such as transportation. This can be achieved through the adoption of personalized ventilation and air cleaners. In previous research activities [2], we investigated the effectiveness of a newly designed personal air cleaner in reducing the indoor airborne transmission of respiratory pathogens. The device was conceived to be placed on desks, creating a protected volume in which the user can breathe clean air. This concept is herein extended to transportation, proposing a device to be installed above the user's head and creating a clean zone where they can breathe safely. Presented in the next sections are the concept of device and the preliminary results from CFD analyses.

2 METHODS

The design of the personal air cleaner is realized by means of 3D CFD analyses, assuming an incompressible, unsteady and turbulent flow. Turbulence was modelled using the unsteady Reynolds-averaged Navier-Stokes (URANS) approach, closing continuum equations with the Realizable k–ε model. The centre and right pictures in Figure 1 show the schematic of the device. It samples the particle-laden air from the environment, purified with an electrostatic filter, and reintroduces clean air through an annular section at its bottom. The internal geometrical configuration ensures the development of a diverging conical air shield, protecting the user (supposed to be sitting below the diffuser) from the surrounding environment. To numerically design the air cleaner, the blue-coloured computational domain in the left picture of Figure 1 was employed. The domain comprises both the inside of the device, to determine the internal air flow patterns, and an external cylinder with a diameter of 3m and 1m high, to assess its fluid dynamics behaviour in the immediate surroundings.
A velocity of 0.6 m/s was set as boundary condition at the inlet of the device.

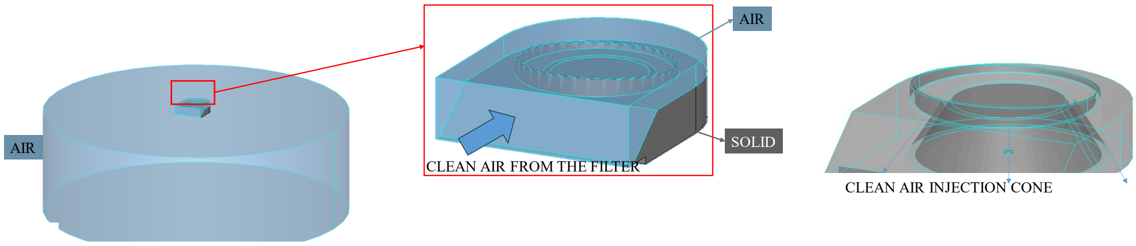

Figure 1. Schematic of the air cleaner (centre and right images) and computational domain employed in numerical simulations (left image).

3 RESULTS AND DISCUSSION

Figure 2a displays a clipped view of the hexahedral-based unstructured grid employed in the simulations, composed by 7,747,639 cells. Figure 2b, on the other hand, shows the predicted mean velocity field (on the same y-z slice of Figure 2a). CFD results show that the device creates quite symmetrical diverging jets, a bit deflecting owing to their mutual influence. They extend for 40 cm along the vertical direction and effectively create a protected inner control volume, where the user can breathe safely.

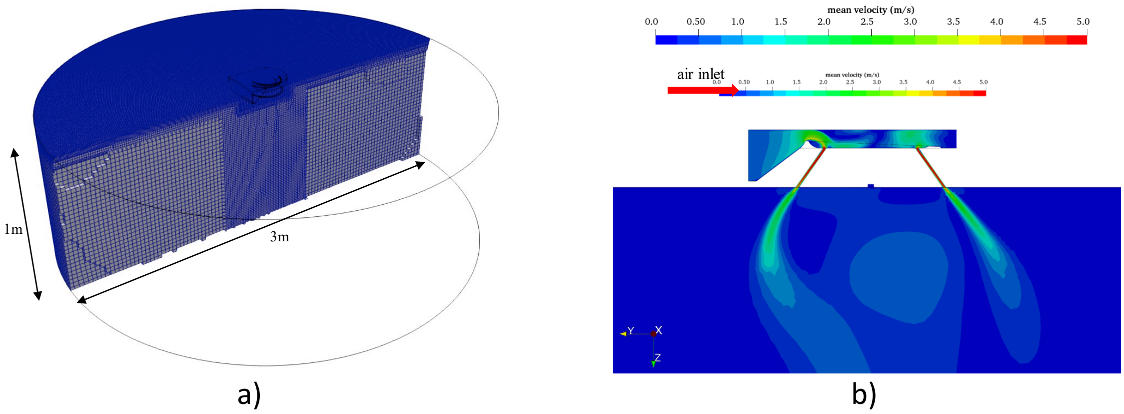

Figure 2. Pictures showing a) the computational grid employed for CFD simulations, b) the predicted mean velocity field on a selected y-z slice.

5 CONCLUSIONS

This paper presents the preliminary results of the CFD design of a novel device aimed at reducing the airborne transmission of respiratory pathogens in transport microenvironments. Additional numerical analyses are being performed, to determine the optimal velocity at the inlet section of the device. Moreover, an experimental campaign is currently underway, in order to validate the numerically predicted velocity fields and to test the filtration efficiency of the proposed device.

6 REFERENCES

[1] Arpino, F., Grossi, G., Cortellessa, G., Mikszewski, A., Morawska, L., Buonanno, G., Stabile, L., **2022**. Indoor Air 32.

[2] Cortellessa, G., Canale, C., Stabile, L., Grossi, G., Buonanno, G., Arpino, F., **2023**. Building and Environment 235, 110222.