Displacement ventilation to avoid COVID-19 transmission through offices

Omar Osman 1 · Mervat Madi 1,2 · Efstratios L. Ntantis 3 · Karim Y. Kabalan 4

Received: 16 February 2022 / Revised: 14 April 2022 / Accepted: 7 June 2022 / Published online: 25 July 2022
© The Author(s) under exclusive licence to OWZ 2022

Abstract
Respiratory infections such as COVID-19 can be spread by respiratory droplets with a diameter larger than 5–10 μm or by droplet nuclei with a diameter smaller than 5 μm. Besides wearing masks, fresh air should be supplied frequently in closed rooms to avoid infections. Constructing and operating new isolation rooms require time, money, and maintenance cost, which are scarce in the current pandemic and in many communities. Displacement ventilation may be a feasible and secure option in temporary hospitals and other buildings to control the disease. This paper investigates using CFD simulations how displacement ventilation systems can deliver high air quality, and thermal comfort and minimize the risk of COVID-19 infection in enclosed spaces.

Keywords CFD · COVID-19 · Displacement ventilation · Enclosed room · Thermal comfort

List of symbols

| Symbol | Description |
|--------|-------------|
| AHU    | Air Handling Unit |
| CFD    | Computation Fluid Dynamics |
| DV     | Displacement Ventilation |
| IAQ    | Indoor Air Quality |
| NS     | Navier Stokes |
| PPD    | Predicted Percentage of Dissatisfied |
| PPE    | Personal Protective Equipment |
| RANS   | Reynolds-averaged Navier-Stokes |
| μ      | Dynamic viscosity (kg/m s) |
| μt     | Turbulent viscosity coefficient (kg/m s) |
| ρ      | Fluid density (kg/m³) |
| Ω      | Mean rotation-rate tensor [rad/s] |
| Si,j   | Mean strain-rate tensor (1/s) |
| U      | Velocity scale of the flow (m/s) |
| l      | Length scale of the flow (m) |
| T      | Temperature (K) |
| P      | Pressure (Pa) |
| τ      | Viscous stress tensor (Pa) |
| σ      | Cauchy stress tensor (N/m²) |
| c_p    | Specific heat capacity at constant pressure [J/(kg K)] |
| u      | Velocity vector (m/s) |
| q      | Heat flux by conduction (W/m²) |
| qT     | Heat flux by radiation (W/m²) |
| Pz     | Maximum of people estimated to occupy the zone under normal condition [persons] |
| a_p   | Thermal expansion coefficient (1/K) |
| Q      | Heat sources other than viscous dissipation (W/m³) |
| Q_turb | Viscous dissipation |
| qoz    | Occupants and electronic devices load (W) |
| ql     | Lighting load (W) |
| q_ex   | Heat conduction load (W) |
| qT     | Total heat conduction load (W) |
| Q_o    | Required volume of outdoor air (L/s) |
| Rp     | Outdoor airflow rate per person (L/s/person) |
| RA     | Outdoor airflow rate required per unit area (L/sm²) |
| Pz     | Maxi number of people estimated to occupy the zone under normal conditions (persons) |
| Az     | Zone floor area (m²) |
| Etz    | Ventilation efficiency of the air distribution system in the zone |
| QDV    | Required air to satisfy the sensible cooling load in a DV system (L/s) |
| thf    | Temperature difference from head to foot level (°C) |

© Springer
1 Introduction

In December 2019, an outbreak of viral pneumonia in Wuhan, Republic of China, led to the discovery of Coronavirus disease. COVID-19’s epidemiology has recently become clearer, as incident cases continue to increase and researchers refine estimates of the disease’s magnitude, transmissibility, and affected populations. The spread of Coronavirus has caused all sorts of fear, worry, and concern in societies all over the world, especially among healthcare workers who are at a higher risk of infection [1, 2]. More than 5.76 million deaths have been confirmed worldwide [3], the global economy is on the brink of collapse, and the variants of COVID-19 are still spreading. The current virus is primarily transmitted between people through respiratory droplets and contact routes, according to the World Health Organization [3]. When people become infected with respiratory disease, their rate of expiratory events rises, increasing the generation and dispersion of virus-carrying droplets. Coughing, sneezing, and talking is also examples of exhalatory activities [4]. Wearing the recommended PPE, such as respiratory protection, protective clothing, protective barriers, and improving personal hygiene are only some of the few physical containment methods that have been adopted [3].

Since people in modern societies are spending most of their time indoors, the infections can spread faster [5]. In addition, because of the broad range of toxins and contaminants found in these enclosed spaces, IAQ has become a major concern [6]. IAQ has been shown to have a major impact on human health, comfort, and productivity on numerous occasions [7, 8]. Dilution of indoor pollutants and contaminants is a common approach to regulate and improve IAQ and decrease the virus-carrying droplets in the air [9].

DV is an air distribution technology that brings air into a zone from outside at a very low velocity. This method of air distribution provides occupants with fresh air and eliminates many heat-source-related pollutants, thus creating a pleasant atmosphere by increasing the IAQ [10]. A standard DV system supplies air through a low induction air diffuser from an AHU. The diffuser is often wall-mounted at floor level and exits the air above the occupied zone from the ceiling vents [11]. The incoming cool air accelerates, spreads over the floor in a thin layer, and due to the heat exchange with heat sources (e.g., occupants, electronic equipment, lights) heats up, becomes less dense, rises to the ceiling, and exits from the exhaust ducts [10].

The most known and used ventilation systems are mixing and displacement ventilation. To determine whether displacement ventilation or mixing ventilation is better suited for each situation we must understand the difference between them. A conventional mixing system usually introduces cool supply air at high velocities at or near ceiling level, thus it randomly mixes the supply air with the room air. Consequently, the Covid-19 droplets will remain in the room. On the other hand, in displacement ventilation, the air is introduced at the floor level and there is minimal mixing with the room air, however, fresh air is continually supplied in the room and minimizes the spread of the covid-19 droplets in the occupied zone. Moreover, the droplets are pushed upwards (Fig. 1).

Because of their significant impact on indoor contaminant removal effects, indoor airflow characteristics are critical for ventilation systems. By solving NS equations and a series of species transport equations, airflow simulation using CFD modeling tools can predict the airflow and indoor contaminant concentration. In many research papers involving parametric studies of room airflow and contaminant dispersion, CFD proved to be very powerful and effective [12–16]. The primary benefit of CFD is that it can offer comprehensive flow data across the entire computing domain, and the simulation performance can be viewed in a variety of ways, including useful descriptions of field distributions and the effects of involved parameters.

Many research papers [17–22] study DV systems as HVAC systems and provide a comprehensive overview of design processes and methods, advantages, limitations, performance criteria, and case studies. In addition, several recent works represented the ventilation application as a solution to decrease the COVID-19 infection in closed rooms [23, 24]. However, this paper focuses on DV and how it can be used as an application to reduce the probability of virus transmission in offices. The work objectives to deliver include:

- Mathematical modeling of the turbulence flow and heat in fluids physics
- Validation of the mathematical model

![Fig. 1 Difference between mixing ventilation (left) and displacement ventilation (right) [36]](image-url)
• Mesh independent test
• CFD simulation of a DV system in office rooms with specific dimensions, design conditions, and cooling loads
• CFD simulation of coughing scenarios to investigate the circulation of droplets particles coming out from occupant’s mouth standing or seated in an office room equipped with a DV system.

2 Mathematical model

2.1 CFD Modeling

CFD is increasingly being used to estimate building airflows providing the benefit of giving airspeed and temperature data at any point. The process of building a CFD model and evaluating the findings has become considerably less labor-intensive because of the recent advancements in computing power, time, and cost [25–28]. With COMSOL software, a k-omega (k-ω) turbulence flow and heat transfer in fluids physics will be solved in addition to non-isothermal multi-physics. With COMSOL software, a k-omega (k-ω) turbulence flow physics, heat transfer in fluid physics, and non-isothermal multi-physics will be solved in The Particle Tracing for Fluid Flow physics using the Finite Element Method (FEM).

2.1.1 Turbulence modeling

Turbulence is a flow field feature characterized by a 3D domain field, a wide range of length and time scales, effective mixing, and highly rotational. The Reynolds number measures the potential of an isothermal flow becoming turbulent [26]. The NS equations are used for turbulent flow simulations. Different turbulence modeling requires different assumptions about predicted fluctuations, resulting in varying degrees of accuracy for distinct flow situations. RANS models are used in this module to predict the influence of the fluctuations on the mean-flow quantities. The Wilcox k-ω model equations are used in this module to predict the influence of the fluctuations on the mean-flow quantities. The Wilcox k-ω model equations are used for non-isothermal multi-physics will be solved in The Particle Tracing for Fluid Flow physics using the Finite Element Method (FEM).

\[
\frac{\partial k}{\partial t} + \rho \mathbf{u} \cdot \nabla k = P_k - \rho \beta \omega + \nabla \cdot \left( (\mu + \sigma \mu_T) \nabla k \right)
\]

\[
\frac{\partial \omega}{\partial t} + \rho \mathbf{u} \cdot \nabla \omega = \frac{\alpha}{k} P_k - \rho \beta \omega^2 + \nabla \cdot \left( (\mu + \sigma \mu_T) \nabla \omega \right)
\]

2.2 Turbulence flow, K-w

The major advantage of using the k-omega (k-ω) model over the k-epsilon (k-ε), is that the former is more accurate in estimating the rate at which jets spread [29]. The k-ω model solves for the turbulent kinetic energy k, and the turbulent kinetic energy dissipation rate \( \omega \) which is also known as the specific dissipation rate. The Wilcox k-ω model equations are given below in Eq. 2 [29]. For steady-state problems, the time derivatives vanish because the temperature does not change with time.

\[
\frac{\partial k}{\partial t} + \rho \mathbf{u} \cdot \nabla k = P_k - \rho \beta \omega + \nabla \cdot \left( (\mu + \sigma \mu_T) \nabla k \right)
\]

\[
\frac{\partial \omega}{\partial t} + \rho \mathbf{u} \cdot \nabla \omega = \frac{\alpha}{k} P_k - \rho \beta \omega^2 + \nabla \cdot \left( (\mu + \sigma \mu_T) \nabla \omega \right) \quad (2)
\]

where:

\[
\alpha = \frac{13}{25} ; \beta = \beta_0 f_\beta ; \beta^* = \beta_0^* f_\beta ; \sigma = \frac{1}{2} ; \sigma^* = \frac{1}{2} ; \mu_T = \frac{k}{\omega}
\]

\[
P_0 = \frac{1}{2} \left( 1 + 70 X_{\omega} \right) \left( \frac{1}{1 + 80 X_{\omega}} \right) ; X_{\omega} = \frac{\Omega_{ij} \Omega_{jk} S_{kl}}{(\rho_0^\omega)^2} ; f_\beta = \frac{9}{100}
\]

\[
f_\beta = \begin{cases} 1 & x_k \leq 0 \\ \frac{1 + 70 X_{\omega}}{1 + 80 X_{\omega}} & x_k > 0 \end{cases}
\]

\[
\Omega_{ij} = \frac{1}{2} \left( \frac{\partial n_i}{\partial x_j} - \frac{\partial n_j}{\partial x_i} \right) ; S_{ij} = \frac{1}{2} \left( \frac{\partial n_i}{\partial x_j} - \frac{\partial n_j}{\partial x_i} \right)
\]

\[
P_k = \mu_T \left( \nabla \mathbf{u} : \left( \nabla \mathbf{u} + (\nabla \mathbf{u})^T \right) - \frac{2}{3} (\nabla \cdot \mathbf{u})^2 \right) - \frac{2}{3} \rho k \nabla \cdot \mathbf{u}
\]

2.3 Heat transfer in fluids

This paper uses the Heat Transfer in Fluids COMSOL Interface for the following Eq. 3 solved [30]. For a steady-state problem, the time derivatives also vanished because the temperature remains constant with time.

\[
\rho C_p \left( \frac{\partial T}{\partial t} + \mathbf{u} \cdot \nabla T \right) + \nabla \cdot (\mathbf{q} + \mathbf{q}_r)
\]

\[
= \alpha_p T \left( \frac{\partial p}{\partial t} + \mathbf{u} \cdot \nabla p \right) + \tau : \nabla \mathbf{u} + Q \quad (3)
\]

2.4 Non-isothermal flow coupling

Fluid flows having non-constant temperatures are referred to as non-isothermal flows. When the temperature of fluid changes, its material characteristics, such as density and viscosity, vary as well. Since in the current applications the turbulent fluid flow properties depend on the temperature the “Non-isothermal Multiphysics” will be used to couple between the main physics; Turbulence flow K-w, and Heat.
transfer in fluids. Referring to the equations section in COMSOL, the two-coupling terms are as follows in Eq. 4:

\[
Q_{vd} = \tau : \nabla \mathbf{u} + Q_{turb} - n_\mathbf{q} = \rho C_p u T_w - T T_+ \tag{4}
\]

### 2.4.1 The particle tracing for fluid flow

The Particle Tracing for Fluid Flow physics is used to calculate particle motion in a background fluid. Drag, gravity, electric, magnetic, and acoustophoretic factors can all influence particle motion. In the current case, the main force that influences the fluid particle is the drag force due to the massless weight of particles and the presence of the displacement ventilation in the room. Newton’s second law in Eq. 5, says that the net force on a particle is equal to its linear momentum’s temporal rate of change in an inertial reference frame.

\[
\frac{d}{dt}(m_p \mathbf{v}) = F_D + F_g + F_{ext} \tag{5}
\]

\[ F_D \text{ is the Drag Force is defined as:} \]

\[
F_D = \frac{1}{\tau_D} m_p (u - v)
\]

### Case 1: Office Room Load and Inputs Calculations

For regular DV applications, the following design steps process provides a simplified way to determine the ventilation rate and supply air temperature based on ASHRAE standards [10, 11, 19]. While the overall building load keeps constant, the calculations determine the airflow needs to maintain the set-point in the room space.

#### Table 1 Design characteristics of the case study

| Design considerations   |   |
|-------------------------|---|
| Occupants               | 2 |
| Set-Point               | 22 °C |
| Floor Area              | 12 m² |
| Exterior Wall           | 9 m² |
| Volume                  | 36 m³ |

### 2.5 Room design

The case study of the current paper is an office room with dimensions of 3 m wide, 4 m long, and 3 m height, equipped with a list of items as shown in Fig. 2 and design characteristics as given in Table 1.

### 2.6 Design considerations

When installing a DV system, one of the most important factors to consider is thermal comfort. ASHRAE Standard 55–2004 specifies the maximum mixture of velocity and temperature in the occupied zone, the PPD due to draft, and the stratification in space [31]. According to ASHRAE Standard 55–2004 and ASHRAE Research Project-949, to maintain thermal comfort the allowable temperature difference between the head and foot of an occupant should not exceed 3 °C [10, 31]. Table 2 lists the assumptions taken for the examined case study [11]:

The following Eq. 6 is the breakdown of loads to determine the total cooling load \( q_T \) in the office.

\[
q_{oc} = (2 \text{People} \times 75W) + 60W = 210W
\]

\[
q_l = 12m^2 \times 25W/m^2 = 300W
\]

\[
q_{ex} = 9m^2 \times 15W/m^2 = 135W
\]

\[
q_T = 210 + 300 + 135 = 645W \tag{6}
\]
Table 2  Design assumptions of the case study

| Loads                                      | Value          |
|--------------------------------------------|----------------|
| Occupant                                   | 75 W per person|
| Lighting (T8 fluorescent lighting)         | 25 W/m²        |
| Computer (CPU and LCD Monitor)             | 60 W           |
| Exterior Wall Conduction (3 × 3)           | 15 W/m²        |

For moderately busy office work applications, ASHRAE Standard 62.1–2004 specifies a 0.3 L/s m² outdoor airflow rate per unit area, Ra, and 2.5 L/s outdoor airflow rate per person, Rp. However, the zone air distribution effectiveness Ez, for DV is considered to be 1.2 [32].

### Determination of the airflow rate to meet the cooling load, QDV:

\[
Q_{DV} = \frac{0.295q_{oe} + 0.132q_l + 0.185q_{ex}}{ho c_p \Delta t_{hf}} = \frac{0.295(210) + 0.132(300) + 0.185(135)}{(1.2)(1.007)(3)} = 34.9 L/s
\]

### Determination of the fresh air flow rate, Qaf:

\[
Q_{af} = \frac{R_p P_z + R_A A_z}{E_z} = \frac{(2.5)(2) + (0.3)(12)}{1.2} = 7.2 L/s
\]

### Determination of the total supply air volume, Qs:

\[
Q_s = \max(Q_{DV}, Q_{af}) = 34.9 L/s
\]

### Calculation of the supply air temperature, ts:

\[
t_s = t_{sp} - \Delta t_{hf} - \frac{A_z q_l}{0.584 Q_s^2 + 1.208 A Q_s} = 22 - 3 - \frac{49}{0.584(34.9)^2 + 1.208(12)(34.9)} = 12.6
\]

### Determination of the return air temperature, te:

\[
t_e = t_s + \frac{q_T}{1.208( Q_s )} = 12.6 + \frac{645}{1.208(34.9)} = 27.9°C
\]

### Adjustment for new supply temperature, QDV:

\[
Q_{DV} = \frac{q_T}{1.208(t_e - t_{sp})} = \frac{645}{1.208(27.9 - 17)} = 48.6 L/s
\]

Table 3  Boundary Condition of the case study

| Boundary Type     | Expression                  | Value           |
|-------------------|-----------------------------|-----------------|
| Turbulence Flow, K-w Boundary Conditions |                            |                 |
| Inlet 1           | Fresh air inlet             | Velocity 2 m/s  |
| Inlet 2           | Fresh air inlet             | Velocity 2 m/s  |
| Outlet 1          | Exhaust (outlet)            | Pressure = 0 Pa |
|                   | (Normal flow)               |                 |
| Wall              | All remaining walls         | No slip conditions|
| Heat Transfer in Fluid Boundary Conditions |                            |                 |
| Temperature 1     | Fresh air inlet             | Supply air      |
|                   | Temperature = 17 °C         |                 |
| Outflow 1         | Exhaust (outlet)            | Outflow         |
| Boundaries heat source 1 | Occupants Boundaries | 60 W            |
| Boundaries heat source 2 | PC Boundaries               | 70 W            |
| Boundaries heat source 3 | One External wall (3 × 3)  | 15 W/m²         |
| Boundaries heat source 4 | Ceiling wall               | 300 W           |
| Thermal Insulation 1 | All remaining walls        | Thermal Insulation |

The supply temperature (ts) should be minimum 17 °C or higher.

### 2.7 Selection of the diffusers design and dimensions

In order to fulfill comfort standards, traditional displacement diffusers are restricted to 0.2 m/s face velocity in normal commercial applications. A diffuser face area of 0.245 m² is required with a supply airflow of 49 L/s and a face velocity of 0.2 m/s.

\[
A_{diff} = \frac{Q_{DV}}{V_{diff}} = \frac{0.049}{0.2} = 0.245 m^2
\]

### 2.8 Boundary conditions

The boundary conditions of the cast study are given in Table 3.

#### Case 2: Coughing in an Office Room

In this case, the design inputs of Case 1 remain the same, however, a constant flow rate is added, representing the coughing flow rate coming out from the mouth of an occupant, i.e., an office room visitor. Referring to cough dynamics [33] the cough lasts 500 ms and the peak that the flow rate of coughing reached is ~ 5 L/s. In the current study, the cough is taken to last for 0.5 s with a constant flow rate of 0.2 L/s
or 0.0002 m³/s, generating 275 massless particles of 10 μm diameter which refers to the largest Covid-19 droplets’ diameter. However, due to the high computational cost of the time-dependent simulation and the negligible effect of the coughing on the airflow rate of the entire room, the simulation is divided into two studies:

1. Steady-state study: the office is simulated after adding a constant airflow of 0.2 L/s that came out from the occupant’s mouth with a temperature of 37 °C.
2. Time-dependent study: The “Particle Tracing for Fluid Flow” physics is applied to mimic the discharge of virus caused by coughing. After the solution of the airflow in the room, this physics traces the particles. The coughing was assumed to generate 275 air particles of 10 μm diameter and the cough lasted for 0.5 s. The massless particle formulation for this investigation is chosen since the masses of bacteria are very small.

### 3 Model Validation

Despite the high availability of experimental data in the literature, very few contain accurate details that can be employed for flow information, thermal boundary conditions, and thermal parameters in space. Therefore, an error analysis must be specified in the experimental data, in order to validate the
current model. Due to previously mentioned lack of experimental data in the literature, the current research paper refers to the validation from COMSOL, available on “COMSOL Multi-physics® Application Library” [34] (see Fig. 3), which has a high similarity with the current model study especially by using the same physics and equations.

Figure 4 refers to the validation of the DV module by comparing experimental data measured from the center of the jet [35], with the simulation data at the same line position. It can be noticed that while the simulation result reflects the general pattern of decreasing temperature with height, the model indicates an over-prediction. This error refers to bad insulation of the test chamber [35] or the absence of buoyancy-induced production of the k-ω model equations.

4 Mesh independent test

The current model tested with four different mesh sizes, reaching the maximum computational power of the system. The mesh sizes are the following:

- Mesh 1: consists of 41,552 domain elements, 4770 boundary elements, 743 edge elements
Fig. 9 Air temperature distribution in Celsius, in the office room (orthographic projections)

Fig. 10 Streamline distribution in the office room with temperature color-bar in Celsius

Fig. 11 Velocity distribution in m/s, in the meeting room (slice view)

Mesh 2: consists of 82,818 domain elements, 7720 boundary elements, 930 edge elements
Mesh 3: consists of 190,160 domain elements, 14,612 boundary elements, 1335 edge elements
Mesh 4: consists of 385,627 domain elements, 22,466 boundary elements, 1665 edge elements

Fig. 12 Pressure distribution in Pascal, in the office room

The mesh of the model in Fig. 5, is made with a red cut line between the points (3,1,0) m and (3,1,3) m, as shown in Fig. 6. The temperature gradient along the red line is plotted and illustrated in Fig. 7, after the end of every modeling test. Due to the fact that the maximum temperature difference between Mesh 3 and Mesh 4 is less than 1 °C and the computational demands for the Mesh 4 is higher, Mesh 3 is selected as a default, for the following simulations.

5 Discussion

The Displacement ventilation system removes Covid-19 droplets since the direction of droplets velocity is chosen as the worst-case scenario to be normal to the vertical axis, yet the ventilation system manages to drive them upwards to the exit. The particles are only brought into the room through coughing which is shown in scenarios A and B. After running the CFD simulation, the following plots are achieved for different cases.

Case 1: Office Room
The CFD plots of air temperature in Fig. 8 show how the DV divides the office into different sections; mainly the upper
hotter zone with a maximum of 27 °C, and a minimum of 19 °C in the lower cooler. The temperature distribution on different planes in Fig. 9 pictures the plume of occupants and equipment, e.g., lights, computers, etc. It is interesting to mention from the orthogonal projection in Fig. 9 that the temperature of the air is 18 at the inlets, and reaches 28°C at the exhaust vanes. In addition, the temperature color bars in Figs. 8 and 9 show how the seated person would experience a conditioned air temperature from 21.5 to 23.4 °C, and a standing person from 21.3 to 24.1 °C. Therefore, the temperature difference between the head and the foot level for a seated occupant would be less than 2 °C, and for a standing one less than 3 °C. The streamlines in Fig. 10 amplify the form of the safe occupied zone by suppling fresh cold air from the diffuser gates. In Fig. 11, the velocity distribution is illustrated in slices and indicates how the conditioned air in blue color is present all over the office with a very low velocity of 0.05 m/s and with no high variations. Finally, Fig. 12 measures the pressure difference which is indicated not to be significantly high. Conclusively, with data received from the plots of temperature distribution, velocity, and pressure difference, the occupants in the office with a DV system would...
be within the thermal stratification comfort condition set by ASHRAE [19, 32, 33].

Case 2: Meeting Room
The following CFD plots illustrate the air circulation in a meeting office, which is a bigger in size room compared to the office room of Case 1. Starting with the axonometric projections in Fig. 13, the DV manages to split the room space into different iso-surfaces with minimum and maximum temperatures, 17 °C, and 25 °C respectively. The temperature distribution in Fig. 14 illustrates the thermal plumes of the occupants, lights, LCD, and projector. Figure 15 visualizes the temperature streamlines and shows the form of the safe occupied zone provided by the air supply from the outside through the diffuser vanes located very close to the floor level. The color bars in Figs. 13 and 14 determine for a seated occupant the air temperature difference would be from 20.4 to 22.4 °C, and for a standing occupant from 20.3 to 23.5 °C. Therefore, the temperature difference between the head and foot would be 2 °C for the seated, and 3.2 °C for standing. The schematic in Fig. 16 shows vertical slices of the conditioned airflow distribution and the bar on the side determines the very low air velocity, around 0.04 m/s with no high variations. Finally, Fig. 17 illustrates in iso-surfaces the very small pressure change. In addition to the temperature and velocity plots in Figs. 13 and 16, the occupants in the meeting room will be provided with thermal stratification comfort conditions within the conditions set by ASHRAE [19, 32, 33].

Case 3: Occupant Cough
The third case examines the possibility of an occupant coughing or forced exhalation, and how the exhaled breath particles will affect the quality of the air environment in the room. Due to the fact that the occupant may be seated or standing, case three shall be divided into two possible scenarios, A and B. In this section, the transmission probability factor is used which is a factor generated using COMSOL. This factor shows the probability of particles passing through a selected surface, boundary, or domain. COMSOL calculates this factor by dividing the total number of particles by the number of particles passing through a boundary or a domain. In this paper, the transmission probability factor is used to show the leaving probability of COVID-19 particles (generated by coughing) from the room through displacement ventilation exits.

5.1 Scenario A: seated occupant cough
The first scenario examines the potential effects of a seated occupant who starts coughing. The execution of CFD simulations resulted in the following figures and plots. Figure 18 shows the temperature difference from the mouth of the occupant to the ceiling vanes, which is determined 3 °C. In addition to Fig. 18, Figs. 19 and 20 traces the trajectory of particles from different projection views. The color bars show the increase of the velocity of the particles, which is 10% higher while rising to the top of the room. Figure 21 shows the total number of the particles on the PC boundaries, that is in the first 500 ms. The transmission probability against time in Fig. 21 illustrates the time the particles take to stick on PC surface boundaries and to exit vanes boundaries. Figures 22, 23 and 24 prove that once a seated occupant cough, 96% of
the particles stick on the PC boundaries after 2.86 s, and after 115.2 s the rest 4% exited from the ventilation exhaust vanes.

### 5.2 Scenario B: standing occupant cough

Scenario B studies the coughing process of a standing occupant in the office, for example of a visitor. The Figures and plots presented below are extracted from CFD simulations. Starting Fig. 25, it shows the temperature increase of the particles from the mouth of the visitor to the top of the ceiling diffuser, which is 4.5 °C; 1 °C higher than in Scenario A. Comparing the minimum temperature of the two Scenarios, it can be observed that it is higher for the standing person because when a person stands the body burns more calories, thus the breath is hotter and this is reflected higher exhaled
Fig. 24 Transmission probability of the particle at the exit boundaries versus time.

Fig. 25 Coughing streamline scenario 2 with temperature color bar in Celsius.

Fig. 26 Coughing streamline scenario 2 with velocity color bar in m/s.

breath temperature. Figures 26 and 27 visualize the trajectory of the particles, with a color bar indicating the velocity increase along the particle’s path. The particles in Scenario B are found to move faster by 5% compared to Scenario A, due to the heavier breath of the standing person who exhales faster as the heartbeat is higher. The transmission probability against time in Figs. 28 and 29 shows that once a seated occupant cough, 96% of the particles stick on the PC boundaries after 2.86 s, and the rest 4% exited from the ventilation exhaust vanes after 115.2 s.
6 Conclusions

Since constructing and operating new isolation rooms requires time, money, and maintenance human resources, which are scarce in the current pandemic and in many communities, DV besides PPE may be a feasible and secure option to control the transmission of COVID-19 virus in buildings. In order to investigate the DV airflow in closed rooms, numerical models of “Turbulence Flow, k-ω” and “Heat Transfer in Fluids” physics with their non-isothermal coupling, were employed, obtaining the following conclusions:

- CFD is a tool to optimize and increase the efficiency of DV systems by visualizing the movement of virus droplets coming out from people by coughing, speaking, and sneezing.
- DV systems can be used as a healthcare system to minimize the enclosed respiratory infections, especially during the current COVID-19 pandemic.
- Case studies demonstrated how a DV system supplied occupied zones with fresh cool air and allowed the hot air that might contain bacterial droplets to exit, maintaining at the same time the thermal comfort of the occupants.
- Case studies showed how fast a DV system eliminates cough droplets that might contain viruses.

Data Availability The data that support the findings of this study are available from the corresponding author upon reasonable request.

Declaration

Conflict of Interest The authors declare that they have no conflict of interest.

References

1. Cowling BJ, Aiello AE (2019) Public health measures to slow community spread of Coronavirus disease. J Infect Dis 221(11):1749–1751
2. Tan Z, Phoon PHY, Zeng LA (2020) Response and operating room preparation for the COVID-19 outbreak: a perspective from the National Heart Centre in Singapore. J Cardiothorac Vasc Anesth 34(9):2331–2337
3. World Health Organization, Coronavirus Disease (COVID-2019) Situation Reports, 2020 https://www.who.int/emergencies/diseases/novel-coronavirus-2019/situation-reports/ (Accessed Feb 9, 2022).
4. Morawska GR, Johnson ZD, Ristovski M, Hargreaves K, Mengersen S, Corbett CYH, Chao YL, Katoshevski D (2009) Size distribution and sites of origin of droplets expelled from the human respiratory tract during expiratory activities. J Aerosol Sci 40(3):256–269
5. da Silvia GM (2020) An analysis of the transmission modes of COVID-19 in light of the concepts of indoor air quality. University of Coimbra, Portugal
6. Zhang J (2020) Integrating IAQ control strategies to reduce the risk of asymptomatic SARS CoV-2 infections in classrooms and open plan offices. Sci Technol Built Environ 26(8):1013–1018
7. Dorgan CB, Dorgan CE, Kanarek MS, Willman AJ (1998) Health and productivity benefits of improved indoor air quality. ASHRAE Trans 104:658–666
8. Sujanová P, Rychtarikova M, Sotto Mayor T, Hyder A (2019) A healthy, energy-efficient and comfortable indoor environment, a review. Energies 12(8):1414
9. Javed S, Ørnes IR, Dokka TH, Myrup M, Holoss SB (2021) Evaluating the use of DV for providing space heating in unoccupied periods using laboratory experiments, field tests and numerical simulations. Energies 14(4):952
10. Price Industries Limited 2016, Engineering Guide - DV.
11. Chen Q, Glicksman LR, Yuan X, Hu S, Yang X (1999) Final report for ASHRAE RP-949: Performance evaluation and development of design guidelines for DV. Cambridge, MA: Department of Architecture, Massachusetts Institute of Technology
12. Du L, Yang C, Dominy R, Yang L, Hu C, Du H, Li Q, Yu C, Xie L, Jiang X (2019) Computational fluid dynamics aided investigation
13. Cole EC, Cook CE (1998) Characterization of infectious aerosols in health care facilities: an aid to effective engineering controls and preventive strategies. Am J Infect Control 26:453–464
14. Jiang Z, Haghighat F, Chen Q (1997) Ventilation performance and indoor air quality in workstations under different supply air systems: a numerical approach. Indoor Built Environ. 6(3):160–167
15. Jiang Z, Chen Q, Haghighat F (1995) Airflow and air quality in large enclosures. J SolEnergy Eng 117(2):114–122
16. Haghighat F, Jiang Z, Zhang Y (1994) Impact of ventilation rate and partition layout on VOC emission rate: time-dependent contaminant removal. Int J Indoor Air Quality Climate 4(4):276–283
17. Kosonen R, Melikov A, Mundt E, Mustakallio P, Nielsen PV (2017) REHVA Guidebook No:23: DV, REHVA: Brussels, Belgium
18. Skistad H, Mundt E, Nielsen PV, Hagström K, Railio J (2002) REHVA Guidebook No.1: DV in Non-Industrial Premises, REHVA: Brussels, Belgium
19. Chen Q, Glicksman L (2003) System performance evaluation and design guidelines for DV, American Society of Heating, Refrigerating, and Air-Conditioning Engineers, Inc.: Atlanta, GA, USA
20. AEC (2006) DV Design Guide: K-12 Schools. Final Report for Public Interest Energy Research Program, California Energy Commission; Architectural Energy Corporation: Sacramento, CA, USA
21. Nielsen PV (1993) DV: theory and design. Aalborg University. Department of Building Technology and Structural Engineering. Aalborg, Denmark
22. Shouhong K, Tian S, Xiangyi M (2015) Comparison of DV, mixing ventilation and underfloor air distribution system. In: International conference on architectural, civil and hydraulic engineering, November 28–29, Guangzhou, China
23. Kermani A (2015) CFD modeling for ventilation system of a hospital room. In: Proceedings of the 2015 COMSOL Conference, Boston, USA
24. Bhattacharyya S, Dey K, Paul AR, Biswas R (2020) A novel CFD analysis to minimize the spread of COVID-19 virus in hospital isolation room. Chaos, Solitons Fractals 139:110294
25. Anderson DA, Tannehil JC, Pletcher RH, Munipali R, Shankarm V (1997) Computational fluid mechanics and heat transfer, 4th edn. CBC Press, Ottawa
26. Stephen BP (2000) Turbulent flows. Cambridge University Press, Cambridge
27. Blazek J (2005) Computational fluid dynamics: principles and applications. Elsevier, Amsterdam
28. Pope SB (2000) Turbulent flows. Cambridge University Press, Cambridge
29. Paul DB, Stuart NL, Robert IF (2005) Computational fluid dynamics. Wiley, New York
30. Wilcox DC (1998) Turbulence Model for CFD, 2nd ed., DCW Industries, La Canada
31. Bird RB, Stewart WE, Lightfoot EN (2007) Transport phenomena, 2nd edn. Wiley, New York
32. ASHRAE (2004) Standard 55–2004—Thermal environmental conditions for human occupancy. American Society for Heating, Refrigerating and Air Conditioning Engineers, Atlanta
33. ASHRAE (2004b). Standard 62.1–2004—Ventilation for acceptable indoor air quality. Atlanta, GA: American Society for Heating, Refrigerating and Air Conditioning Engineers
34. Gupta JK, Lin CH, Chen Q (2009) Flow dynamics and characterization of a cough. Indoor Air 19(6):517–525
35. Mazoni D, Guitton P (1997) Validation of DV simplified models. In: The 5th international building performance IBSPA conference. September 8-10, Prague, Czech Republic. Vol (I), pp 233-239
36. Displacement ventilation vs. mixing ventilation. SimScale. (2020, October 12). Accessed April 13, 2022, from https://www.simscale.com/blog/2017/12/displacement-ventilation-cfd/

Publisher's Note Springer Nature remains neutral with regard to jurisdictional claims in published maps and institutional affiliations.