Materials Today: Proceedings 56 (2022) 2967–2972

Contents lists available at ScienceDirect

Materials Today: Proceedings

journal homepage: www.elsevier.com/locate/matpr

Computational fluid dynamics-based disease transmission modeling of SARS-CoV-2 Intensive Care Unit

Shivam Prajapati [a,*], Nishi Mehta [b], Aviral Chharia [c,d], Yogesh Upadhyay [e]

[a] Department of Mechanical Engineering, National Institute of Technology, Agartala, TR 799046, India
[b] Mechanical Engineering Department, Sardar Vallabhbhai National Institute of Technology, Surat, GJ 395007, India
[c] Mechanical Engineering Department, Thapar Institute of Engineering and Technology, Patiala, PB 147004, India
[d] Computer Science and Engineering Department, Thapar Institute of Engineering and Technology, Patiala, PB 147004, India
[e] Department of Mechanical Engineering, Zakir Husain College of Engineering & Technology, AMU, India

ARTICLE INFO

Article history:
Available online 9 November 2021

Keywords:
COVID-19
Aerosol
CFD
HVAC

ABSTRACT

Covid-19 has become one of the most severe diseases causing acute respiratory problems and has killed millions of people worldwide. It was declared as the ongoing pandemic by the World Health Organization. It is an infectious virus which can be transmitted by sneezing, coughing and exhalation of air by any infected person. There are certain places having high chances of becoming contaminated like hospital rooms. In this context, we studied the transmission of Covid-19 particles in an ICU room. We have considered the combined effect of both of air-conditioning (AC) and ceiling fan in the room. The infected person can transmit the disease when under influence of fan and AC. The work highlights the flow of aerosol particles considering the combined effect as well as the individual effects of fan and AC. The results also emphasized that the aerosol particle flow have a promising application in sanitizing the room.

Selection and peer-review under responsibility of the scientific committee of the 3rd International Conference on Contemporary Advances in Mechanical Engineering

1. Introduction

Novel coronavirus (caused by the SARS-CoV-2 virus) has infected around 96.2 M people across the world and has caused the death of around 2.06 M people. During the infected persons' coughing, sneezing or exhaling, the COVID-19 virus gets transmitted through the droplets generated by them. These droplets don't hang in air, as they are too heavy. Hence, they just fall on any surfaces or walls [1].

Aerosols are the microfluidic particles which are produced when someone sneezes or coughs. The consideration of shape, size as well as evaporation kinetics of these particles are necessary as these properties of particles have a significant impact on the transport of aerosol as well as droplets in the system [2]. It has been noted that the disease can be transmitted through large droplets (>5µm) capable of carrying sufficient viral load produced by coughing and sneezing (Gralton et al. 2011, World Health Organization 2004). The forces acting on a particle primarily depend on particle size and its position in the flow field [2]. Especially, while considering the case of airborne transmission of the diseases, the risk of infection of these diseases increases significantly for the persons inhaling the viruses which are attached to tiny respiratory droplets (usually, less than 5 mm, therefore considered as aerosols), which can be produced by any normal action like normal breathing, talking, etc., and are suspended in the air for a significant time duration in the order of hours as well as they can be accumulated in poorly ventilated indoor spaces [3,4].

In the previous studies researchers have done detailed study on classrooms, gyms, lifts and restaurants (Jiarong Hong et al. 2020) etc. but there has been no proper computational analysis for the ICU room where there are greater chances of aerosol transmission because of the various infected patients visiting the ICU room. Abuhegazy et al. [2] numerically investigated the aerosol transmission related to COVID-19 in a class room. They investigated transport of aerosol and surface deposition using computational fluid dynamics. They explored the particle size in different locations of the room like windows and glass barriers. They found that a significant fraction (24%–50%) of particles smaller than 15 µm exit the system within 15 min through the air conditioning system. Particles larger

* Corresponding author.
E-mail address: prajapatishivam64@gmail.com (S. Prajapati).

https://doi.org/10.1016/j.matpr.2021.11.013
2214-7853/
Selection and peer-review under responsibility of the scientific committee of the 3rd International Conference on Contemporary Advances in Mechanical Engineering

Nomenclature

Symbol	Meaning
ρ	Density (kg/m^3)
ρ_d	Density of particle (kg/m^3)
t	Time (s)
$\overrightarrow{V}$	Velocity vector (m/s)
P	Pressure (Pa)
$\overrightarrow{S}$	Source term
μ	Dynamic viscosity (m^2/s)
T	Temperature (K)
k	Turbulent kinetic energy (J)
c_P	Specific heat capacity (J/kg K)
S_T	Total Source Term
x_i	Cartesian direction of i
u_i	Velocity component of i (m/s)
x_j	Cartesian direction of j
α_k	Effective inverse Prandtl number for the turbulent kinetic energy
α_e	Effective inverse Prandtl number for the turbulent kinetic energy's dissipation
μ_{eff}	Effective dynamic viscosity (m^2/s)
G_k	Generation of turbulent kinetic energy
ε	Turbulent dissipation rate
S_k	User defined source term
S_ε	User defined source term
R_ε	Source term for Renormalization
$C_{l\varepsilon}$	Model constant = 1.42
$C_{2\varepsilon}$	Model constant = 1.68
V_d	Particle velocity (m/s)
F_D	Coefficient of Drag force
$\overrightarrow{g}$	Gravitational acceleration (m/s^2)
F_L	Saffman Lift Force (N)
F_B	Brownian Force (N)
d	Particle's diameter (m)
C_C	Cunningham coefficient
F_D	Coefficient of Drag Force
$\dot{m}$	Mass flow rate of particles (kg/m^3s)
r	Radius of droplets (m)
n	Number density of droplets (m^{-3})
K	Fluid thermal conductivity (W/m K)

than 20 μm almost entirely deposit on the ground, desks, and nearby surfaces in the room. Asymptomatic people transmitted COVID-19. A study on this aerosol transmission of COVID-19 particles was presented by Liu et al. [5]. A large eddy simulation solver combined with some other numerical methods was used by them for simultaneously resolving some important but complex indoor processes like the effects of turbulence, interplay of flow and aerosol, thermal effect, effect of filtration, etc. which were caused mainly due to the air conditioners.

Crawford et al. [6] presented a study on aerosol transmission of airborne pathogens in an ICU. It is believed that viruses containing aerosols produced by any infected patient can enter into other rooms through ventilation and put other medical staff at risk. It was observed in the results that the usage of High Flow Nasal Cannula in the oxygenation technique modified the air flows which were generated due to coughing as well as normal breathing. This further increased the infectious airborne particles' shedding. The schlieren optical method was used in these experiments. This study also uses a 3D Computer Fluid Dynamics model based on a Lattice Boltzmann Method to simulate the air flows as well as the movement of numerous airborne particles produced by a patient's cough within an ICU room under negative pressure. Mutuku et al. [7] presented a study on transmission and airflow of aerosol transmission in human airways. The differences in the deposition efficiencies as well as the deposition patterns of toxic as well as the pharmaceutical aerosols in both the conditions of human airways, which were healthy as well as obstructed were investigated in detail by them. The transmission of airborne virus in confined spaces like elevators was investigated by Dhouk et al. [8]. They considered an elevator as a prototyping model of a confined space and showed the detailed study of the design of ventilations. They also investigated about the air circulation through multiphase CFD and performance of air purifier in an elevator. The disease and mortality severity were increased because of the accumulation of the virus in enclosed spaces. It remained airborne in such spaces which caused the upper respiratory tract's repeated exposure as well as high viral load's buildup in it.

Bhattacharyya et al. [19] measured the effectiveness of aerosol sanitizer mixed air from air conditioner used to disinfect the room having COVID-19 virus. They concluded that sanitizer can be distributed effectively by utilizing the high turbulent fields which were created in the room. The CFD analysis model used in this study was the transition SST k-ε model. Mirzaie et al. [20] analyzed the effect of installing transparent barriers in front of classroom seats in order to prevent the effect of coughing or sneezing of a COVID-19 infected person standing in the front area of the classroom. It was observed that the barriers were quite effective for preventing the infection. Moreover, increment in air velocity of ventilation which was in the direction of airflow proved to be beneficial as it decreased the time which droplets took to get trapped by the partition. If the partitions were not present, then the persons whose seats were the closest to the infected persons got affected the most as the droplet concentration was the highest in that area. Many other studies have been carried out implementing the CFD analysis in areas having COVID-19 infected persons.

It is observed that no detailed study has been done till now on ICU rooms for aerosol transmission of Covid-19. ICU rooms are the indoors places where there is much chance of transmission but with better ventilations and settings this infection of aerosol infection of virus could be minimized. Because of the accumulation of the virus in enclosed spaces, the probability of contracting to the disease as well as the amount of severity and the risk of mortality gets significantly increased.

Hospitals come under essential services so it is high time that researchers should focus more on analyzing the flow behavior of the aerosol transmission by some robust computational method. This paper deals with simulation for aerosol transmission of Covid-19 particles in an ICU room. The room has been analyzed and divided on the basis of R factors. The room has been divided into three types of zones: (i) Red zone (ii) Yellow zone and (iii) Green zone. The red zone has a maximum chance of getting contaminated. The room is designed in such a way to get least contamination of the Covid-19.

2. Proposed methodology

The CAD model of ICU consists of a bed, a table, a chair, regular sized fan, almirah, air conditioner consisting of inlet and outlet. These entities have been considered for the CFD work. However, this model can be utilized even for other geometries of the infrastructure in the room. The assembly of the study room was created in Solidworks and later it was imported in the Design modeler of commercial ANSYS FLUENT 2019 R3. The top base of the fan has been considered as outlet and lower base is considered as inlet of the fan. An aero solid model was created in ANSYS DESIGNMODELER after importing the SOLIDWORKS 2016 geometry. Air volume is extracted from the SOLIDWORKS model in order to generate the grid. Various control volumes were made by dividing fluid volumes. The dimensions of various room entities are shown in Table 1.

The computational domain is discretized using mesh generation. The analytical solutions obtained by Navier-Stokes equations are valid only for ideal condition flows. Real flows can be analyzed through numerical methods having equations modified with algebraic approximations. The Discretization of governing equations is done by dividing the entire spatial domain into numerous small finite control volumes by using the meshing process. Integration of governing equations over each control volume is performed by discretely conserving quantities like energy, mass, momentum, etc. over each control volume. An extremely large no. of cells in the domain might increase the computation time and a very few cells might result in inaccuracy of results. ANSYS MESHER is used for meshing. Mode toolbar or switch-to-solution mode command is used to transfer the meshed figure into the solution mode. In the solution mode, one can perform functions like boundary condition setting, defining the properties of fluid, execution of the solution, viewing as well as the post processing of the results.

Grid is generated from different control volumes consisting of a bed, a table, regular sized fan, almirah, air conditioner as shown in Fig. 1. The meshes are generated using tetrahedral as well as hexahedral elements (refer Fig. 3). The intricate parts of the control volumes of the model were filled using the hexagonal mesh elements, whereas the remaining parts were tetrahedral mesh elements. The governing equations which were used on each element were the mass, momentum as well as the energy conservation equations. Moreover, the Finite Volume Method was used in this CFD analysis. The number of nodes in the mesh were 63,390 and the number of mesh elements were 2,20,107. Boundary conditions are an integral part of any CFD analysis. The inlet boundary conditions were considered for the Fan as well as the AC inlet. The inlet air velocity, mass flow rate and the temperature for the AC were considered to be 3.1 m/s, 0.569625 kg/s and 288.15 K respectively. The outlet mass flow rate for AC was considered to be 0.569625 kg/s, the same as the inlet. The inlet conditions for the fan were 3.3 m/s as inlet air velocity, 7.155 kg/s as air mass flow rate, and 303.15 K as inlet temperature. The room temperature was considered to be 303.15 K. The outlet of the fan as well as AC were considered to be the pressure outlets.

The assumptions made for the simulation are given as follows:

- The air conditioning and ceiling fan systems in selected bedrooms are fully functioning and running well (refer Fig. 4)
- The room is fully sealed and enclosed without any holes or gaps (excluding doors, windows and exhaust vent).
- The outside temperature on the surface of the room is constant.
- Internal heat emitted from the digital devices and lights will be neglected due to minimal effect on the temperature.

Inflation layers are layer (s) of fine mesh which are created in the domain's area of interest (refer Fig. 2). Here, the inflation layers are created by selecting the wall boundary. They are used to capture flow at the boundaries. The transition ration used in the meshing was 0.247 with the growth rate 1.2. The Fig. 2 shows the inflation layer of the mesh.

Table 1
Dimensions of the room entities.

Room Entity	Size
Room Domain	$5\ m \times 5.5\ m \times 3\ m$
Fan diameter	$1.5\ m$
AC Inlet	$1\ m \times 0.3\ m$
AC Outlet	$1\ m \times 0.2\ m$

Discretization is required by a governing equation's numerical solution. Boussinesq approximation along with second order upwind schemes are used to serve this purpose [9]. The flow physics of this system is analysed using unsteady CFD analysis. Modelling for laminar-transitional flow is done for this system of the ICU room. For serving this purpose, the RNG $k-\varepsilon$ model is used. Yakhot et al. [10] developed this model by using Re-Normalization Group (RNG) methods for renormalizing the Navier-Stokes equations for applying to the small-scale motions to know their effects. A single turbulence length scale is used to determine the eddy viscosity in the standard $k-\varepsilon$ model, due to which, the turbulent diffusion is calculated only at the particular specified scale. The reality is however, not exactly the same, as motions of all the scales have their contribution in the turbulent diffusion. The production term is changed in the epsilon equation's modified form in the mathematical technique of the RNG approach for the derivation of the turbulence model similar to that of the standard $k-\varepsilon$ model.

2.1. Governing equations

Ventilation air flow modelling:

For incompressible steady air flow, the conservation equations of mass, momentum and energy are given as:

$$\frac{\partial \rho}{\partial t} + \nabla \cdot \left(\rho \vec{V}\right) = 0 \tag{1}$$

$$\rho\left(\frac{\partial \vec{V}}{\partial t} + \vec{V} \cdot \nabla \vec{V}\right) = -\nabla P + \mu \nabla^2 \vec{V} + \vec{S} \tag{2}$$

$$\rho \frac{\partial T}{\partial t} + \rho \vec{\nabla} \cdot \left(T\vec{V}\right) = \nabla\left(\frac{k}{c_P}\nabla T\right) + S_T \tag{3}$$

Turbulence modelling:

In order to simulate the airflow in indoor environments, the RNG k-ε turbulence model was considered to be quite suitable [10,11]. The turbulent kinetic energy k as well as the dissipation rate ε transport equations are given as:

$$\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_i}(\rho k u_i) = \frac{\partial}{\partial x_j}\left[\alpha_k \mu_{eff} \frac{\partial k}{\partial x_j}\right] + G_k - \rho\varepsilon + S_k \tag{4}$$

$$\frac{\partial}{\partial t}(\rho \varepsilon) + \frac{\partial}{\partial x_i}(\rho \varepsilon u_i) = \frac{\partial}{\partial x_j}\left[\alpha_e \mu_{eff} \frac{\partial \varepsilon}{\partial x_j}\right] + C_{l\varepsilon}\frac{\varepsilon}{k}(G_k) - C_{2\varepsilon}\rho\frac{\varepsilon^2}{k} - R_\varepsilon + S_\varepsilon \tag{5}$$

Discrete phase modelling:

The particle trajectory analysis was done in the air flow, having dilute concentration of droplets in the present case. Newton's second Law in Lagrangian setup [12–14] Fis used for evaluating the virus-carrying droplets' trajectories. Following are the motion equations for it:

$$\frac{dV_d}{dt} = F_D\left(\vec{V} - \vec{V}_d\right) + \frac{\vec{g}(\rho_d - \rho)}{\rho_d} + F_L + F_B \tag{6}$$

$$F_D = \frac{18\mu}{d^2 \rho_d C_C} \tag{7}$$

The Saffman lift force as well as the Brownian force [15] are given in Eq. (6), F_D being drag force, C_C being the Cunningham Coefficient [16,17]:

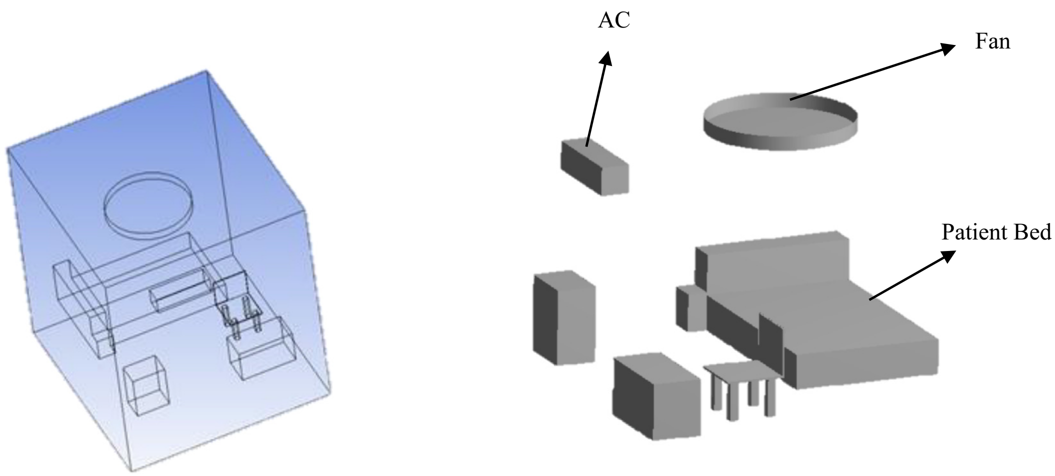

Fig. 1. (a) Room geometry (b) Room Entities.

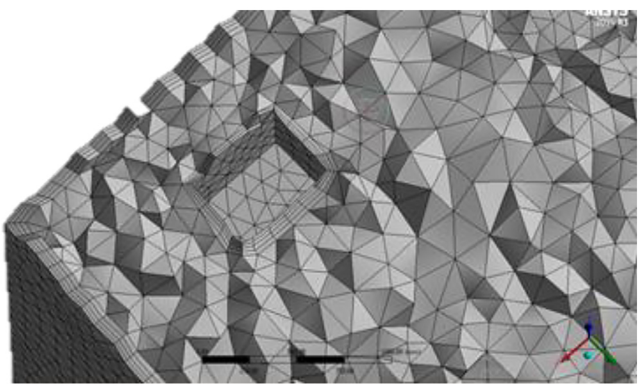

Fig. 2. Inflation Layer.

$$C_C = 1 + \frac{2K}{d}\left(1.257 + 0.4e^{-\left(\frac{1.1d}{2k}\right)}\right) \tag{8}$$

Following is the formula for particles' mass flow rate:

$$\dot{m} = \frac{\left(\frac{4}{3}\Pi r^3\right) \times \rho_d \times n}{t} \tag{9}$$

Three different grid sizes having number of elements as 220107, 246765 and 298978 were considered. The simulation was performed for all three grid sizes computational domain. It was observed that the results were converging for grid 1 and grid 2. So, the grid size which was least computationally expensive was considered for all the numerical simulations.

3. Results and discussion

The COVID-19 pandemic has demonstrated the extraordinary transmissibility of the virus. Our model integrates innovative in situ measurements and CFD simulations to offer a systematic assessment of the hazards of airborne virus transmission caused by asymptomatic persons inside a COVID-19 dedicated ICU. Our observational results allow us to characterize disease transmission patterns in great detail. Many significant outcomes are highlighted and established by the air transmission pattern and streamlining presented concerning an ICU room. This section elaborates the obtained results from the performed experiments and discusses the results. The simulation is conducted by using the ANSYS Fluent CFD software.

The fan's velocity curve is shown in Fig. 5(a). In the case of COVID-19, the flow from the fan has the ability to disseminate the disease and transmit the virus around the room if the infected patient is lying on the bed. It is clear from the diagram that various velocities exist in different areas of the room. The speed ranges from 0 to 4.06 m/s. Moreover, there are some regions where velocity is below 0.5 m/s. Ventilation, in particular, can assist in disseminating virus-containing particles to wider regions outside the

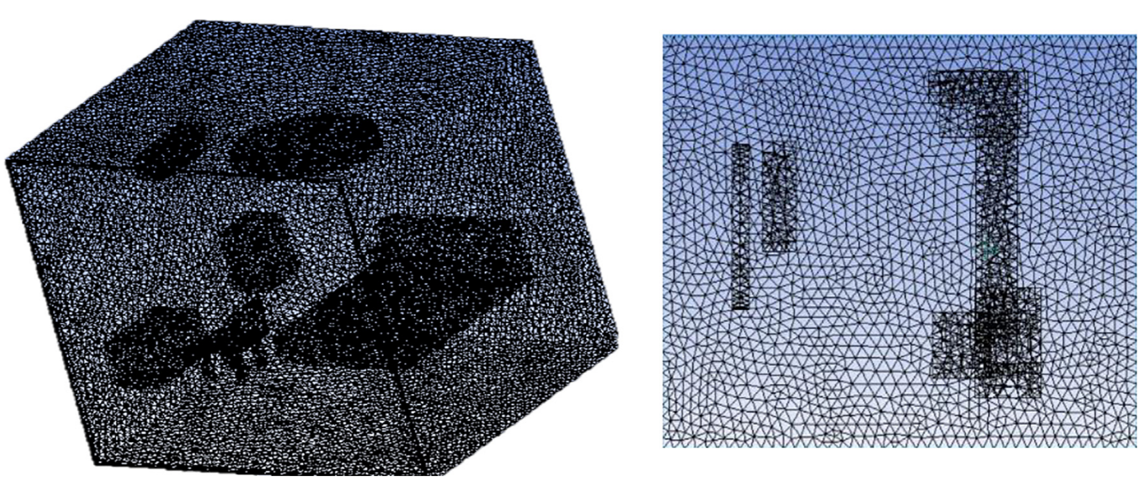

Fig. 3. Mesh.

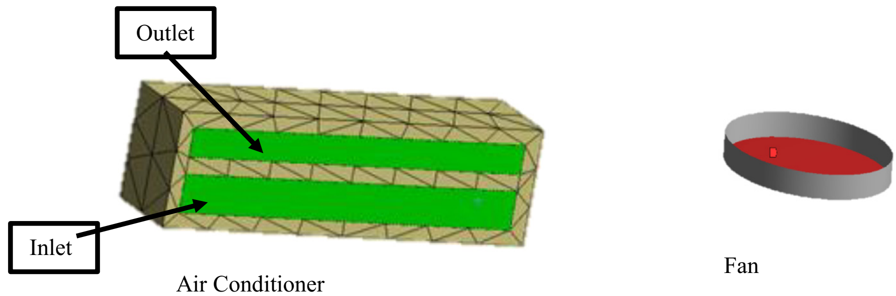

Fig. 4. Different components used in ICU Room.

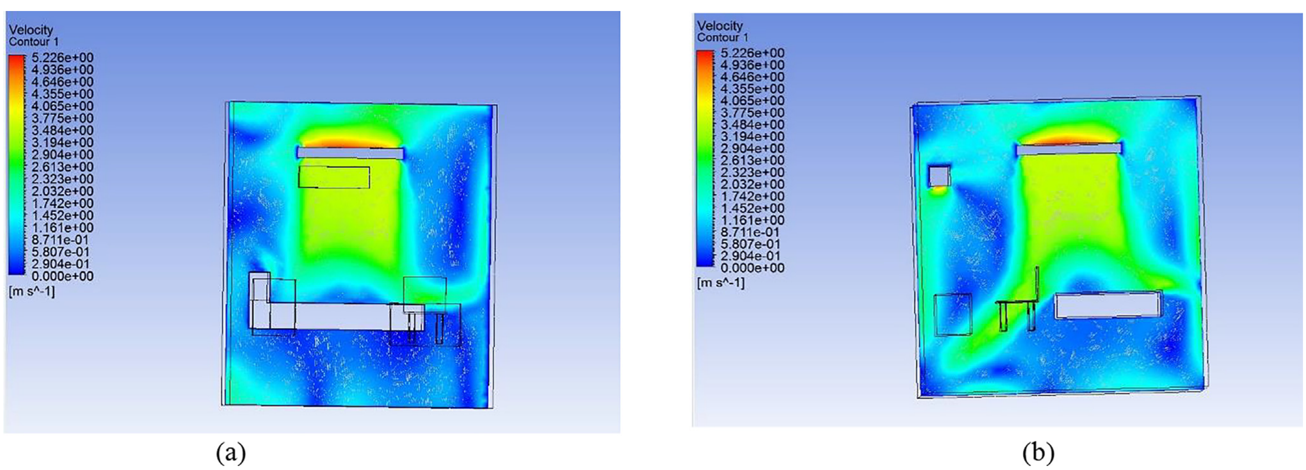

Fig. 5. (a) Velocity contours of the air flow through fan showing aerosol transmission. (b) Velocity contours of combined effect of Fan and AC.

proximity of asymptomatic persons, even though it allows virus-containing particles to be removed. Depending on the relative location of the particle emitter, ventilation, and space settings, improper ventilation can potentially lead to local hot spots with hazards that are orders of magnitude higher than other areas.

Fig. 5(b) shows the disease transmission in a COVID-19 dedicated ICU having both Air conditioner and Fan. The combined effect of both the Air Conditioner and Fan are taken into account while solving the model. Fig. 2 shows the regions with a velocity below 0.5 m/s and regions with a velocity higher than 0.5 m/s. This shows that the above room as more chances of disease transmission since rooms with AC have low to no ventilation. Fig. 3. Shows the velocity streamline from the fan inlet in the COVID-19 ICU room.

Fig. 6 illustrates the streamlines developing from the top of the ICU room. The streamlines show the path of virus transmission inside the room. If a non-infected person/medical staff falls directly into the path of the streamlines, he would have a high chance of disease contamination. The streamlines can be seen descending downhill in these figures, creating an influence on the patient beds and the isolation chamber floor. Following that, the flow striking the floor bounces back and spreads to the walls. Thus, airflow can increase particle deposition on surfaces, resulting in patched areas with significant surface contamination, as evidenced by the enormous quantity of COVID-19 RNAs recovered from hospital floors and air vents [12]. It is also worth noting that these faceted particles and droplets that land on surfaces may form highly resilient microscale residues, which may shield viruses from environmental changes and contribute to long-term surface infectivity reported [18].

When the ceiling ventilation is located in the rear corner of the ICU Room (i.e., far from the emitter), particles are dispersed throughout the room. The dispersion of particles is mainly restricted to the region before the front side of the room. The area around the vent, in particular, can pose a considerable risk, with paramedics present at a hot spot in the back inhaling several times more particles than those present at a safe position. When the ventilation is moved to the emitter side, the risk is considerably decreased compared to the previous situation.

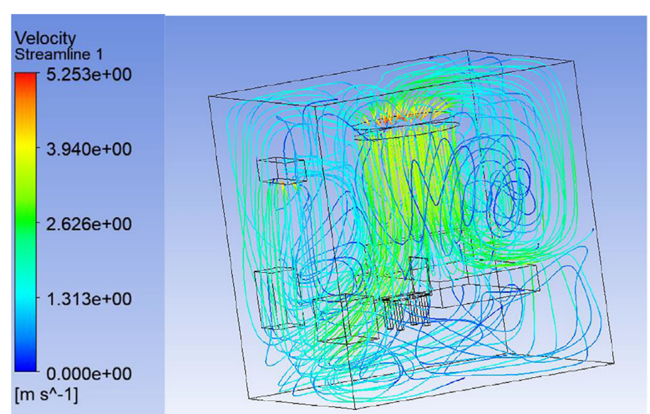

Fig. 6. Streamlines showing the combined effect AC and Fan.

The total particle number passing through this location during the simulation time, which can be interpreted as the number of particles a person can inhale during the simulation time, is used to calculate the risk of a person encountering virus-containing particles in one specific location (R).

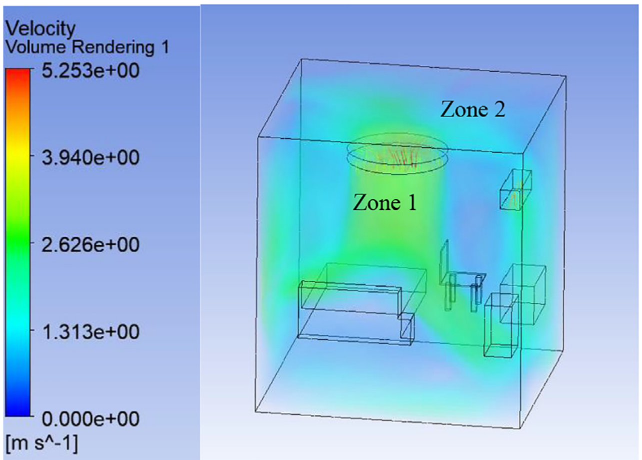

Fig. 7. Volume rendering of the aerosol transmission showing various zones.

The wall contours in the 3D contour plots are the risk contours that have been spatially averaged in the x, y, and z axes, respectively. Furthermore, assuming that each person is 1.75 m tall, from the horizontal slices of the risk contour at various locations, it can be inferred that places lying in direct contact with the plotted streamline are in a high-risk zone as compared to those away from it, i.e., the safe criterion can be calculated based on the distance from the streamlines. It is worth noting that the overall danger defines safe and unsafe zones in each situation.

Fig. 7 demonstrates the volume rendering of the contamination regions in the ICU room. It is observed that the green region i.e. zone 1 which is just below the fan has the maximum chances of a person being contaminated. While light green region i.e. zone 2 has significantly lower chances of contamination. Zone 1 has high chances of getting contaminated. Flow pattern of the air is quite more in the region. So, it is advised not to sit below fan in an ICU room when fan is on. Through this contour, one would be analyse easily where they should give ventilation in a room.

4. Conclusion

The flow characteristics of a regular ICU unit are investigated computationally in this study, which is essential for disinfecting the room air and thereby preserving the lives of physicians, nurses, and healthcare workers. The path of these streamlines should be studied while designing a sanitizing/virus controlling system for the ICU. Because of relatively constant flow circulation zones in the area and the enormous quantity of particle deposition on surfaces, ventilation at a single site, even at the maximum rate currently used, is very ineffective at eliminating particles. The patterns of airflow along in an ICU with ventilation have been investigated in this work. As the contaminated droplets also move with the airflow, the people, especially the medical workers coming in their streamlines have a high probability of getting infected. The work can be positive contribution in designing for HVAC organizations. Further they can make use of streamlines pattern to design the innovative solutions that target the zones according to their impact. A proper air cleaning technology can be implemented that purify air according to the concentration of the zones, with the help of proper sensors.

CRediT authorship contribution statement

Shivam Prajapati: Conceptualization, Methodology, Software, Formal analysis. **Nishi Mehta:** Validation, Writing – review & editing. **Aviral Chharia:** Methodology, Writing – original draft. **Yogesh Upadhyay:** Visualization, Writing – review & editing.

Declaration of Competing Interest

The authors declare that they have no known competing financial interests or personal relationships that could have appeared to influence the work reported in this paper.

References

[1] "Coronavirus disease (COVID-19)." https://www.who.int/emergencies/diseases/novel-coronavirus-2019 (accessed Aug. 03, 2021).
[2] M. Abuhegazy, K. Talaat, O. Anderoglu, S.V. Poroseva, Numerical investigation of aerosol transport in a classroom with relevance to COVID-19, Phys. Fluids 32 (10) (2020) 103311, https://doi.org/10.1063/5.0029118.
[3] J. L. Santarpia *et al.*, "Aerosol and surface contamination of SARS-CoV-2 observed in quarantine and isolation care," *Sci. Reports 2020 101*, vol. 10, no. 1, pp. 1–8, Jul. 2020, doi: 10.1038/s41598-020-69286-3.
[4] S. Zhu, S. Jenkins, K. Addo, M. Heidarinejad, S.A. Romo, A. Layne, J. Ehizibolo, D. Dalgo, N.W. Mattise, F. Hong, O.O. Adenaiye, J.P. Bueno de Mesquita, B.J. Albert, R. Washington-Lewis, J. German, S. Tai, S. Youssefi, D.K. Milton, J. Srebric, Ventilation and laboratory confirmed acute respiratory infection (ARI) rates in college residence halls in College Park, Maryland, Environ. Int. 137 (2020) 105537, https://doi.org/10.1016/j.envint.2020.105537.
[5] H. Liu, S. He, L. Shen, and J. Hong, "Simulation-Based Study on the COVID-19 Airborne Transmission in a Restaurant," *medRxiv*, p. 2020.12.10.20247403, Dec. 2020, doi: 10.1101/2020.12.10.20247403.
[6] C. Crawford *et al.*, "Modeling of aerosol transmission of airborne pathogens in ICU rooms of COVID-19 patients with acute respiratory failure," *Sci. Reports 2021 111*, vol. 11, no. 1, pp. 1–12, Jun. 2021, doi: 10.1038/s41598-021-91265-5.
[7] J.K. Mutuku, W.-C. Hou, W.-H. Chen, An Overview of Experiments and Numerical Simulations on Airflow and Aerosols Deposition in Human Airways and the Role of Bioaerosol Motion in COVID-19 Transmission, Aerosol Air Qual. Res. 20 (6) (2020) 1172–1196, https://doi.org/10.4209/AAQR.2020.04.0185.
[8] T. Dbouk, D. Drikakis, On airborne virus transmission in elevators and confined spaces, Phys. Fluids 33 (1) (2021) 011905, https://doi.org/10.1063/5.0038180.
[9] S. Bhattacharyya, H. Chattopadhyay, R. Biswas, D. R. E. Ewim, and Z. Huan, "Influence of Inlet Turbulence Intensity on Transport Phenomenon of Modified Diamond Cylinder: A Numerical Study," *Arab. J. Sci. Eng. 2019 452*, vol. 45, no. 2, pp. 1051–1058, Nov. 2019, doi: 10.1007/S13369-019-04231-9.
[10] V. Yakhot, S.A. Orszag, S. Thangam, T.B. Gatski, C.G. Speziale, Development of turbulence models for shear flows by a double expansion technique, Phys. Fluids A Fluid Dyn. 4 (7) (1992) 1510–1520, https://doi.org/10.1063/1.858424.
[11] X. Zhao, Q. C.-I. air, and undefined 2019, "Inverse design of indoor environment using an adjoint RNG k-ε turbulence model," *Wiley Online Libr.*, vol. 29, no. 2, pp. 320–330, Mar. 2018, doi: 10.1111/ina.12530.
[12] W. J *et al.*, "Bioinspired shape-memory graphene film with tunable wettability," *Sci. Adv.*, vol. 3, no. 6, Jun. 2017, doi: 10.1126/SCIADV.1700004.
[13] T.N. Verma, A.K. Sahu, S.L. Sinha, Study of Particle Dispersion on One Bed Hospital using Computational Fluid Dynamics, Mater. Today Proc. 4 (9) (Jan. 2017) 10074–10079, https://doi.org/10.1016/J.MATPR.2017.06.323.
[14] T. Dbouk, D. Drikakis, On coughing and airborne droplet transmission to humans, Phys. Fluids 32 (5) (2020) 053310, https://doi.org/10.1063/5.0011960.
[15] A. Li and G. Ahmadi, "Dispersion and Deposition of Spherical Particles from Point Sources in a Turbulent Channel Flow," *http://dx.doi.org/10.1080/02786829208959550*, vol. 16, no. 4, pp. 209–226, 2007, doi: 10.1080/02786829208959550.
[16] Y. Chen, Z. Deng, Hydrodynamics of a droplet passing through a microfluidic T-junction, J. Fluid Mech. 819 (May 2017) 401–434, https://doi.org/10.1017/JFM.2017.181.
[17] C. Zhang, S. Wu, F. Yao, Evaporation regimes in an enclosed narrow space, Int. J. Heat Mass Transf. 138 (Aug. 2019) 1042–1053, https://doi.org/10.1016/J.IJHEATMASSTRANSFER.2019.04.113.
[18] C.K. Yinda, J.R. Port, T. Bushmaker, I. Offei Owusu, J.N. Purushotham, V.A. Avanzato, R.J. Fischer, J.E. Schulz, M.G. Holbrook, M.J. Hebner, R. Rosenke, T. Thomas, A. Marzi, S.M. Best, E. de Wit, C. Shaia, N. van Doremalen, V.J. Munster, K. Subbarao, K18-hACE2 mice develop respiratory disease resembling severe COVID-19, PLOS Pathog. 17 (1) (Jan. 2021) e1009195, https://doi.org/10.1371/journal.ppat.1009195.
[19] S. Bhattacharyya, K. Dey, A.R. Paul, R. Biswas, A novel CFD analysis to minimize the spread of COVID-19 virus in hospital isolation room, Chaos, Solitons & Fractals 139 (2020) p. 110294, https://doi.org/10.1016/j.chaos.2020.110294.
[20] M. Mirzaie, E. Lakzian, A. Khan, M.E. Warkiani, O. Mahian, G. Ahmadi, COVID-19 spread in a classroom equipped with partition–A CFD approach, Journal of Hazardous Materials 420 (2021), https://doi.org/10.1016/j.jhazmat.2021.126587 126587.