Analyzing social distancing policy effectiveness using Computational Fluid Dynamics inside a bus to prevent COVID-19 airborne transmission

A.Pavansai, P.Deepak, S.Ragul Hari, R.Harish*, M.B. Shyam Kumar

School of Mechanical Engineering, Vellore Institute of Technology, Chennai, Tamil Nadu-600127, India

*Corresponding author E-mail: harish.r@vit.ac.in

Abstract:

Coronavirus disease (COVID-19) is a kind of disease that transmits from one body to another through air by the moist particles caused during sneezing and coughing. As a result, to reduce the virus spreading accordingly, 1.83 m of social distancing has been advised to be followed among the humans. Also, it is not well studied that whether the ambient wind and relative humidity (RH) will cause COVID-19 laden cough droplets to transport farther in the air, and make the current social distancing practice ineffective. In this study, computational fluid dynamics simulations are carried out to analyze the transient transport, condensation/evaporation, and deposition of COVID laden droplets emitted by coughs, with the different environmental conditions, cough velocities and Relative Humidity using the developed bus model of size 12 m x 3.5 m x 3.2 m. Different conditions of sneezing and coughing from the human’s i.e., the laminar and the turbulent flows of the laden droplets in the air inside the bus were also examined. The distance between the two virtual humans is set as 6 m for a 12 m bus length in order to track the distance covered by cough particles. The facial covering effect on reducing the airborne transmission of the cough droplets has also been evaluated. It is found that due to the ambient air and humidity, the generation of secondary laden droplets occur which travels far and accumulates on the ground or any other third human being around and causes a strong potential risk to their health. The secondary droplets are transforming into large droplets due to high humidity and the hygroscopic effect is evaluated. The 6 feet social distancing is found to be ineffective for halting the spread of viruses among human beings because the micro-laden droplets caused during the sneezing and coughing are influenced by the convection effects and transport from one body to another within 5 s. It is thus recommended to wear masks for both infected and healthy humans to reduce the airborne cough droplets.

1. Introduction:

The outbreak of the COVID-19 virus creates a massive impact and destruction in all aspects of human well-being. Even the most powerful countries in the world suffer to tackle the effects of the pandemic as there is no proper medication officially released. Governing bodies and United Nations have introduced different measures to protect humans from being prone to the virus, using facemask and sanitizer has been an essential form of living [1]. Different protective measures for individuals like social distancing between humans are being the new normal [2]. A tentative distance of 1.8 m from each other is being advised, to avoid being contagious to viral infection. So, it creates a tough environment for industries working with huge manpower to function with their routine practices. The transportation sector, tourism, and many other sectors are being restricted and affected by the social distancing policy. As people couldn’t afford to be idle for a longer run, they are bound to work for their survival despite the pandemic. This paper will focus
on analyzing the effectiveness of the social distancing policy by carrying out Computational Fluid Dynamics (CFD) simulation on a bus model. A detailed study has been carried out to track the path and the distance traveled by the airborne cough droplets. A transportation model of the bus has been considered, primarily a bluff body design of the bus is created and also placing a victim of COVID-19 virus inside the closed body[3]. By considering different scenarios such as bus being stagnant or moving with different velocities and also including many influencing factors such as inlet air, sneeze, cough velocity, etc. path tracking analysis of pathogens has been obtained and mapped it to graphs and figures.

The research done by Bhattacharyyaa et al. [1] studied on the mixing of air from an air-conditioner with an aerosol sanitizer in an isolation room. The airflow pattern was analyzed such that the sanitizer droplets that are sprayed could reach all the corners of the isolation room. There are numerous studies carried out to analyze the air flow patterns in different ventilation systems in isolated rooms and enclosures [2-4]. This will prevent the infection of people who work in the isolation rooms such as nurses, doctors, health care workers, etc. The study by Dudalski et al. [5] assessed the transmission of airborne viruses using bio-aerosol sampling through a distance of 1m, which was later validated with CFD results. Empirical models and numerical methods were used along with experiments to study the diffusion of droplets in enclosures and are reported in literature [6,7]. There are also numerous experimental [8-10] and analytical models [11-13] developed to estimate the mass flow rates of bidirectional air exchanges in open enclosures [14-17] and rooms.

2. Methodology:

A rectangular passenger bus model, designed in SOLIDWORKS is taken for the analysis. The dimensions and specifications of the bus model are given below.

2.1 Bus dimensions and specifications:

The designed bus model (Figure 1) took has dimensions of 12 m x 3.5 m x 3.2 m and it consists of four windows of dimensions 1.5 m x 0.94 m on both sides of the bus. The windows are separated by a distance of 4.5 m on both sides. Also, the windows are located at a height of 1.96 m from the base of the bus. The material of the bus taken for the analysis is aluminum. The bus compartment is modeled as an empty rectangular box; in actual buses, it is furnished with seats, partitions, and luggage racks, as well as passengers. The doors are closed; in reality, doors are often left open. The effect of wheels is not considered. There are three passengers inside the bus, the infected passenger is at the rear and the other two normal passengers are at the front as shown in the Figure 1. The distance between the infected passenger and the middle standing passenger is 6m and the front passenger is 10m from the infected passenger. The height of the passenger is 1.82m. The windows are symmetrically located and the distance of the center of each window from the front side of the bus is given in Table-1.

Table 1: The distance between center of windows and front end of bus
The whole simulation was done based on turbulent conditions of the air. When the bus cruises at different speeds, the air inlet velocity through windows are different and they follow a different path. To understand this, a simulation similar to a wind tunnel experiment using ANSYS FLUENT under turbulent wind conditions was carried out. The different velocities, with which the bus was traveling are given below in the Table-2.

Table 2: Velocities with which the bus cruises

| S.NO | Velocity of bus (in kmph) | Velocity of bus (in m/s) |
|------|--------------------------|-------------------------|
| 1    | 0                        | 0                       |
| 2    | 20                       | 5.556                   |
| 3    | 30                       | 8.334                   |
| 4    | 40                       | 11.112                  |

The velocities were taken up to 40kmph because there is not much change in the experiment beyond this velocity.
3. Flow visualization of window inlet:

To begin the simulation, the below steps are to be followed:

3.1 Pre-Processing:

This is the primary step of CFD simulation which helps in describing the geometry. A fluid domain is created which encapsulates the bus model. The domain of interest filled with air is then further divided into smaller tetrahedral segments known as the mesh generation. Boolean is used to separate the bus body from the fluid domain.

![Figure 2: Schematic showing the model after meshing](image)

After meshing, the different sections are named as inlet, outlet, walls, and body which in our case is the bus. As the bus travels, the air touches the bus through the inlet and exits through the outlet of the fluid domain. In this process, the air enters into the windows at different velocities which needs to be determined.

3.2 Solver:

After meshing, the model is exported to ANSYS FLUENT and the governing equations, materials and boundary conditions are set. Standard k-epsilon with enhanced wall treatment is taken for the best possible output. Second-order accurate discretization scheme was used for pressure, whereas for momentum discretization, the second-order upwind scheme was used keeping a time step of 0.1 s. The solver was initiated with the different vehicle speeds throughout the domain with 5% longitudinal turbulence intensity. Various boundary conditions employed are as follows:
Inlet: Velocity inlet of 0, 5.556 m/s, 8.334 m/s, 11.112 m/s (With 5% turbulent intensity and turbulent viscosity ratio of 10 and also, the velocities are taken with an interval of 10 kmph to have uniformity as a boundary condition.)

Outlet: Pressure outlet (ambient pressure with 5% backflow intensity)

Domain walls: Velocity same as bus velocity

Bus walls: Stationary

Body surfaces: With open windows (to let the air in)

3.3 Post-Processing:

After the simulation, the results can be observed and analyzed with different contour plots of velocity, temperature, pressure, density, and vector plot of streamlines. Given below are the velocity contours after simulation at different bus speeds. Figure 3 (a), 3 (b) and 3 (c) shows the velocity of inlet air at the windows. A YX plane Z=1 bisects the YZ plane of bus at center represents the velocity distribution across the bus.

Figure 3: Velocity contour at 20kmph (5.556m/s)
The above figures are taken from the simulation of bus model enclosed in a fluid domain with air inside as fluid. It represents the condition of a moving bus. It is similar to wind tunnel experiment but in simulated form. Here, the velocity of air entering into the bus from windows have been measured. Although the exact velocity cannot be found, a non-uniform velocity distribution of air through bus windows have been taken from the given velocity range provided by the velocity contour for study purpose.

In the above Fig 3, the velocity of inlet air through all the windows is the same with velocity in between 0.4-0.86 m/s, in fig 4, the velocity of air through all windows is the same with a range in between 1-1.7 m/s, in fig 5 the velocity of air in the front windows is in between 0-1.22 m/s and the rear windows is
1.22-2.43 m/s. A table has been provided with values that have been taken from velocity contour after the simulation in ANSYS FLUENT.

### Table 3: Boundary conditions for velocity inlets of bus

| S.NO | Velocity of bus (kmph) | Velocity of bus (m/s) | Velocity of air through windows as per results (m/s) |
|------|------------------------|-----------------------|----------------------------------------------------|
|      |                        |                       | Front windows (Win 1, Win 2)                        |
|      |                        |                       | Rear windows (Win 3, Win 4)                         |
| 1    | 0                      | 0                     | 0.2                                                |
| 2    | 20                     | 5.556                 | 0.7                                                |
| 3    | 30                     | 8.334                 | 1.5                                                |
| 4    | 40                     | 11.112                | 1                                                  |

### 4. Discrete particle method:

For the simulation a human being model with a height of 1.82m and the natural velocities at which cough and sneeze particles get released are considered. Sprays can be a little difficult to track, but some scientists have managed to make a rough estimate like naturally, cough particles can travel fast up to 50 mph and can execute almost 3,000 droplets in just one go, and coming to Sneez particles, they can travel up to 100 mph and creates roughly 100,000 droplets. By considering average cough and sneeze particle size, it varies from 0.5-12 μm [3]. In these simulations, a standard value of 4 μm is considered. To show the flow of sneeze and cough particles, injections are given from the injected person's inlet mouth. Cough and sneeze flows enforce high velocities of low viscosity gas through narrow openings; therefore, they are turbulent flows. The present study uses standard k-epsilon turbulence model with enhanced wall treatment. This model gives better results for problems involving ventilation, air quality, and air-borne infection risk in the health care facilities. Since the discrete phase method is employed, its approach is to ensemble large sneeze and cough particles, which are injected in the continuum, and equations for heat and mass transfer are solved simultaneously. Neglecting radiation, the mechanisms for droplet mass and temperature change are convection and evaporation. Sherwood correlations relate mass transfer coefficient to the Reynolds and Schmidt numbers.

\[
m_p c_p (dT_p/\text{dt}) = h_A \left(T_\infty - T_p\right) + (dm_p/\text{dt})h_{g}\text{s} \quad (1)
\]
The trajectory of a discrete phase droplet can be determined by integrating the force balance written in the Lagrangian reference frame. This force balance equates the particle inertia with the forces acting on the droplet. The component form of this equation in the Cartesian coordinate can be written as follows [4].

\[
\frac{du_p}{dt} = F_D \left( u-u_p \right) + g(\rho_f-\rho) + F \quad (2)
\]

The boundary conditions for the discrete phase are as follows:

**Diameter of particles**: 4 μm

**Material taken**: water- liquid (typically what saliva made of)

**Velocity of the particles**: 20, 30, 40 m/s is taken.

**Time period of sneeze**: 4 seconds

**Temperature of particles**: 338 K

**Flow rate**: 2.51e-7, 3.77e-7, 4.96e-7 kg/s (for 20, 30, 40 m/s respectively)

**Injections**: 8 identical injections are taken for flow visualization of particles.

A simple pressure-velocity coupling is considered along with the least-squared cell-based gradient. Second-order discretization is used for pressure and second-order upwind is considered for momentum, turbulent kinetic energy and dissipation rates. Here the gravity is taken into consideration as the particles will reach the surface of the bus. After performing the hybrid initialization with all the boundary conditions given, the calculation is performed by taking 700 numbers of time steps and 0.1 s as the time step size. Also, the maximum iteration/time step is taken as 20. As per the boundary conditions, the particles come out from capsule shaped human mouth at different velocities. Similarly, the bus travels at different velocities which in turn have different velocities of air inlet through windows thus altering the trajectory of human cough and sneeze particles.

5. Results and Discussion:

The main aim of the paper is to track the safest distance inside a bus that a human being needs to maintain. To track the distance, disc-shaped trackers are installed inside a bus with a range of 1.5 m from the rear bus wall and 0.5 m spacing in between two adjacent discs. There are a total of 21 discs that have been installed in the bus. The number of iterations taken is 700 after which, there is no activity observed inside the bus. Next, the results of the simulations for different scenarios are discussed.

Figures 6 and 7 are conditions at rest i.e., bus velocity is equal to 0m/s. The inlet air through windows is taken as 0.2m/s because when the bus is at rest, there will be air entering the windows with some velocities due to disturbances in the surroundings.
Figure 6: Particle residence time at 20 m/s cough velocity and bus at rest

Figure 7: Particle residence time at 30 m/s cough velocity and bus at rest
Since there are disc trackers on the bus floor equally spaced at a distance of 0.5 m, the characteristics of particle tracking have been analyzed from animation video in ANSYS FLUENT and distance travelled by the particles have been observed w.r.t the disc trackers and the distance has been measured by how many discs the particles crossed. From the above figures, the distance traveled by the particles inside the bus due to sneezing and coughing is 6.0m, 6.8m, and 7.5m for the cough velocities of 20m/s, 30m/s and 40m/s respectively. The human cough velocity has been taken with an interval of 10 m/s starting from 20 m/s. The maximum human sneeze velocity or the velocity with which a human being can emit disease causing droplet particles goes up to 44 m/s and any number of simulations can be done within the range of 44 m/s. Because to have uniformity and study purpose, these values has been considered.

In the above case, since it is a stationary condition there is less disturbance from outside and the particles travel farther and many people will have a chance of getting affected. Figures 8, 9 and 10 represents the bus velocity equal to 0 m/s. The air inlets through rear and front windows are taken as 0.2 m/s which we got from previous simulation for the bus. As the human cough velocity increases, the distance travelled by the particles increases.
Figure 9: Particle residence time at 20 m/s cough velocity and bus at 40kmph

Figure 10: Particle residence time at 30 m/s cough velocity and bus at 40kmph
Similarly, the distance has been found with the help of disc trackers and animation video for the particles when the bus is at 40 kmph. From the above figures 11-13, the distance travelled by particles is less than 0.5 m for the human cough velocities of 20 m/s and 30 m/s. Also, for human cough velocity of 40 m/s, the distance found was 0.5 m. This is mainly due to the outside disturbance of air which flows through the windows, resulting in low distance. Here the velocity of air from front and rear windows is taken as 1 m/s and 2.2 m/s which we obtained from previous simulation for bus model.

Also, the distance travelled by particles at 20 kmph and 30 kmph bus velocity has been tracked and displayed in the below table along with previous readings. These are the total simulations done for different bus and human cough velocities in order to find distance covered by disease causing particles.

Table 4: This table represents the Distance w.r.t velocities
| S.NO | Velocity of bus(km/hr) | Velocity of bus(m/s) | Velocity of human cough(m/s) | front inlet (Win 1, Win 2) | back inlet (Win 3, Win 4) | Distance (m) |
|------|------------------------|----------------------|-----------------------------|----------------------------|----------------------------|--------------|
| 1.   | 0                      | 0                    | 20                          | 0.2                        | 0.2                        | 6.0          |
| 2.   | 0                      | 0                    | 30                          | 0.2                        | 0.2                        | 6.8          |
| 3.   | 0                      | 0                    | 40                          | 0.2                        | 0.2                        | 7.5          |
| 4.   | 20                     | 5.56                 | 20                          | 0.7                        | 0.7                        | 1.3          |
| 5.   | 20                     | 5.56                 | 30                          | 0.7                        | 0.7                        | 1.5          |
| 6.   | 20                     | 5.56                 | 40                          | 0.7                        | 0.7                        | 2            |
| 7.   | 30                     | 8.34                 | 20                          | 1.5                        | 1.5                        | 0.5          |
| 8.   | 30                     | 8.34                 | 30                          | 1.5                        | 1.5                        | 0.65         |
| 9.   | 30                     | 8.34                 | 40                          | 1.5                        | 1.5                        | 0.70         |
| 10.  | 40                     | 11.12                | 20                          | 1                          | 2.2                        | <0.5         |
| 11.  | 40                     | 11.12                | 30                          | 1                          | 2.2                        | <0.5         |
| 12.  | 40                     | 11.12                | 40                          | 1                          | 2.2                        | 0.5          |

Figure 12: variation of velocity of the bus w.r.t to pressure inside the bus (image to the left) and distance travelled by the particles(image to the right) distributions

The Figures 13,14 and 15, illustrates the streamline flow in the bus. The streamline flow patterns are plotted and analyzed to determine the particle distribution inside the bus.
Figure 13: Stream line flow when bus at rest

Figure 14: Stream line flow when bus is at 40kmph
6. Conclusion:

Hence the distance that the COVID-19 particles cover inside a moving bus with air disturbance has been measured. To achieve this, a simulation for the bus has been carried out in order to find the velocity of air through the windows of the bus. After finding out the inlet air velocity, the values have been taken to analyze the particle tracking inside the bus with cough particles coming out of human mannequin at different velocities to find the distance travelled by the particles with the help of trackers. Also, it is found that, the safest distance which the humans shall maintain in order to prevent the spread of COVID-19 inside a bus is 7.5m. When the bus velocities are greater than 40kmph the particles will not be able to travel farther due to the high disturbances created by the air through windows which pushes the particles backward, only the passengers sitting or standing behind the infected passenger will get affected. Over this velocity, the safest distance will be around 1 m. When the bus is at rest or having a halt in a bus station, the spread of COVID-19 is greater covering a larger distance and affecting the passengers in that region significantly. So, it is better to reduce halts and set a limited number of bus stations, and is, even more, better to have a single starting and ending destination. It is more hazardous to close the windows and doors when the bus is at rest as the COVID-19 will travel farther due to no disturbances from the air outside.

References:

1) Bhattacharyya, S., Dey, K., Paul, A. R., & Biswas, R. (2020). A novel CFD analysis to minimize the spread of COVID-19 virus in hospital isolation room. Chaos, Solitons & Fractals, 139, 110294. doi:10.1016/j.chaos.2020.110294

2) Jianjian Wei PhD, Yuguo Li PhD, Airborne spread of infectious agents in the indoor environment, American Journal of Infection Control, 44 (2016) S102-S108.
3) Naiping Gao, Jianlei Niu, Transient CFD simulation of the respiration process and inter-person exposure assessment, Building and Environment 41 (2006) 1214–1222

4) Ji-Xiang Wang, Xiang Cao, Yong-Ping Chen, An air distribution optimization of hospital wards for minimizing cross-infection, Journal of Cleaner Production, Volume 279, 2021, 123431, ISSN 0959-6526, https://doi.org/10.1016/j.jclepro.2020.123431.

5) Dudalski, N., Mohamed, A., Mubareka, S., Bi, R., Zhang, C., & Savory, E. (2020). Experimental Investigation of Far Field Human Cough Airflows from Healthy and. Indoor Air. doi:10.1111/ina.12680

6) Deepthi Sharan Thatiparti, Urmila Ghia, Kenneth R. Mead, Assessing Effectiveness of Ceiling-Ventilated Mock Airborne Infection Isolation Room in Preventing Hospital-Acquired Influenza Transmission to Health Care Workers, ASHRAE Trans. 2016 ; 122(2): 35–46

7) Yang J, Sekhar SC, Cheong KW, et al. Performance evaluation of a novel personalized ventilation-personalized exhaust system for airborne infection control. Indoor Air 2015;25:176-87

8) Harish. R, Venkatasubbaiah. K, Mathematical modeling and computation of fire induced turbulent flow in partial enclosures, Applied Mathematical Modelling, vol.37, pp: 9732-9746, 2013.

9) Harish. R, Venkatasubbaiah. K, Transport phenomena of turbulent fire spread through compartment connected to vertical shaft in tall building, Fire Safety Journal, vol.61, pp: 160-174, 2013.

10) Harish. R, Venkatasubbaiah. K, Effects of buoyancy induced roof ventilation systems for smoke removal in tunnel fires, Tunnelling and Underground Space Technology, vol.42, pp: 195-205, 2014.

11) Harish. R, Venkatasubbaiah. K, Numerical investigation of instability patterns and nonlinear buoyant exchange flow between enclosures by variable density approach, Computers & Fluids, vol.96, pp: 276-287, 2014.

12) Harish. R, Venkatasubbaiah.K, Large Eddy Simulation of thermal plume behavior in horizontally partitioned dual enclosure, Building Simulation, vol.8(2), pp: 137-148, 2015.

13) Harish. R, Venkatasubbaiah. K, Numerical study of water spray interaction with fire plume in dual chambers connected to tall shaft, Fire Safety Journal, vol. 74, pp: 1-10, 2015.

14) Harish. R, Venkatasubbaiah. K, Non-Boussinesq approach for turbulent buoyant flows in enclosure with horizontal vent and forced inlet port, Applied Mathematical Modelling vol. 40, pp: 927-941, 2016.

15) Harish. R, Venkatasubbaiah. K, Numerical simulation of turbulent plume spread in ceiling vented enclosure, European Journal of Mechanics-B/Fluids, vol.42,pp:142-158, 2013.
16) Harish. R, Effect of heat source aspect ratio on turbulent thermal stratification in a naturally ventilated enclosure, Building and Environment, 143, pp:473-486, 2018.

17) Harish. R, Buoyancy driven turbulent plume induced by protruding heat source in vented enclosure, International Journal of Mechanical Sciences, 148, pp:209-222, 2018.