Computational study of Corona Virus diffusion in a closed environment

S.Harish, G.N.Nhaarikha, R.Harish*

School of Mechanical Engineering, Vellore Institute of Technology, Chennai, Tamil Nadu 600127, India
*Email address: harish.r@vit.ac.in

Abstract: The biggest challenge that is faced by the human race after World war 2 is the Covid-19, which has affected more than 27.8 million people. The Covid 19 pandemic has entirely changed the livelihood of the people. It is not only seen as a global health crisis but also a Socio-economic issue that has disrupted the Governments worldwide. There is situation where normality is being questioned in the post pandemic era. Many regulations such as Quarantine, self-isolation, social-distancing, and travel restrictions have been advised to prevent the spread of covid-19. After several months of quarantine, the world is now slowly pacing its way back to normality as the vaccine has been invented and brought into use. It is expected that the educational institutions, work places and other public arenas will be opened for the use. However, the regulation of wearing mask and following social distancing are mandatory in public places. Considering the above situation as a need of the hour, this article mainly focuses on the Computational study of the Corona Virus diffusion for a confined space such that it can be applied to places with high population mass. The numerical model was simulated using ANSYS fluent and the results obtained are in excellent accordance with the literature.

Keywords: Covid 19; CFD Simulation; virus diffusion

1. Introduction
The world is now witnessing a global pandemic which has led to the fall of many economies. Throughout the entire world, schools, colleges, factories, and all other public areas have been ghosted due to this outbreak. Many countries are now going through the stage of recovery and people are moving towards following their normal routines with precautionary measures. These measures majorly include the acts of wearing a mask and following social distancing. Many studies have been conducted since the onset of the outbreak regarding the morphology the virus, its spreading pattern, and the preventive measures to be undertaken to avoid spread and infection. One such study analyzed the flow pattern in an isolation room, a room where the virus affected people are isolated in a hospital. It was concluded that the most effective way of controlling the spread of droplets (containing the virus) is by implementing a parallel directional airflow pattern in the isolation rooms.[1] Another study was conducted experimentally and numerically to analyze and understand the physics behind the human sneeze. The sneeze action was modeled numerically to study the droplet motion and its flow behavior. The experiment gave a realistic data which was similar to the numerical data obtained from the CFD model which helped in understanding the sneeze dynamics.[2] Another study was conducted to analyze the respiratory droplet transmission in an office room with only two people inside the room. Distinct types of ventilation strategies like personal ventilation, mixing ventilation and displacement ventilation were
examined in this study. It was concluded that the droplet behavior can vary if the sneeze or cough velocity is very high in the case of personal ventilation.[3] This study opened doors for many ideas on the ventilation systems that can be used in our model. The research done by Suvanjana Bhattacharyya et al.[4] investigated the effective mixing of air from an air-conditioner mixed 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. This will prevent the infection of people who work in the isolation rooms such as nurses, doctors, health care workers, etc. The behavior of cough model was developed by Particle Image Velocimetry (PIV) and Hot Wire Anemometry experiments conducted in different conditions. 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. This research inferred the quantification of air movements caused from coughs to determine the safe distance that has to be maintained between two people to mitigate viral infection. One of the researches was, analyzing the droplet fallout and diffusion or dilution time for a bacterium in the coughed region. The transient analysis simulation provided the path and direction of the particle and its concentration was traced to check the formation zones of the pathogen [6]. Another study stated that the most effective way to prevent pathogen contagion is to cater proper ventilation. Ventilation systems should be designed such that it will fulfill the needs of the respective environment which will be best way in diluting the pathogen concentration.[7] One another research by Ji-Xiang Wang et al.[8] simulated the sneezing process to analyze the virus carrier flow pattern and life span once it is released from patient’s mouth. The mobility of the pathogen and the cross-infection probability were evaluated with the parameters, total maximum time, and overall particle concentration, when a proper outlet facility is provided, the overall particle concentration can be reduced to 95% and the total maximum time by 60% . Another study found that displacement ventilation in a room lowered the chances of exposure to virus from a normal respiration activity. But the probability of cross-infection was high when there was an action of sneeze or cough in a situation when two persons stand facing each other[9]. The studies which were published recently in the domain of Covid virus transmission were mostly in agreement to the ‘2m distance’ requirement to prevent transmission of virus from one person to another. But the current information shows that the virus particle containing droplets can channel through air to another person who is at a distance greater than 2m. Masks are an effective way to prevent the spread. In addition to wearing a mask, maintaining social distance of 2m would create a fall in transmission rates. When an infected person is not wearing a mask, there is a high chance of spread to people who are within a radius of up to 10m [10]. There are also numerous experimental [11-13] and analytical models [14-17] developed to estimate the mass flow rates of bidirectional air exchanges in open enclosures [18-20] and rooms.

This study applies the concepts of computational fluid dynamics to analyze motion and behavior of velocity streamlines and to calculate the distance which people must maintain to avoid the spread of the virus considering the ventilation conditions. Two classrooms are modeled using SOLIDWORKS. The dimensions of first room are taken as 5m x 8m x 3.5m and that of second is taken as 10m x 11.5m x 7m. ANSYS Fluent software was used to create a cough and sneeze model and to evaluate the results. A steady state, 3-D turbulent, multi phase flow using volume of fluids was generated. The plots were created using Tecplot software using the data obtained from Fluent. The findings of this study can be applied to various public places such as school and college classrooms, offices, and many other closed room environments.
Table 1: Thermophysical properties of multiphase materials

| Materials | $\rho$ [kg/m$^3$] | $\mu$ [kg/m-s] | $C_p$[KJ/kg K] | $k$[W/m-K] |
|-----------|-----------------|----------------|----------------|-------------|
| Air       | 1.225           | 0.000017       | 1              | 0.024       |
| Water     | 998.2           | 0.001003       | 4.182          | 0.6         |

2. Methodology

2.1. Geometry description

A practical and real-life example of an educational institution (School class room) is taken for the numerical study. The rooms are in the shape of cuboid with dimensions 5m x 8m x 3.5m and 10m x 11.5m x 7m. Room 1 consists of 6 students, 1 faculty, an entrance and exhaust whereas Room 2 consists of 24 students, 4 fans, 2 entrances and 3 windows. Two locations in room 2 are considered for this study, location 1 is opposite to entrance and location 2 is adjacent to entrance. A numerical simulation is carried out separately for room 1 and room 2 to analyze the virus diffusion when people cough or sneezed by considering factors such as ventilation and results are obtained.

Figure 1: Schematic diagram of classroom 1

Figure 2: Schematic diagram of classroom 2

2.2. Initial and boundary conditions

A pressure based steady state analysis was carried out by neglecting the effect of gravity. The multi phase, flow- turbulence models are turned on during the iterations. Air is chosen as primary phase and water as secondary phase. Turbulence intensity is taken to be 5% with viscosity ratio as 10 for the turbulence model. The solution initialization is hybrid with time step size 0.01 and the reference frame is taken to be ‘relative to cell zone.’ Table 2 and 3 denotes the various initial and boundary conditions of two rooms used to run the simulation.
For room1 the no-slip boundary condition was given to walls with occupant mouth, nose and room entrance were taken to be velocity-inlet and exhaust as pressure-outlet whereas occupant mouth and room entrance were given velocity-inlet, fans as fan-inlet and windows as exhaust-outlet for room2.

### Table 2: Boundary conditions of Room 1

| No. of models | Type            | Magnitude |
|---------------|-----------------|-----------|
| Mouth-Cough   | Velocity-inlet  | 1 m/s     |
| Nose-Sneeze   | Velocity-inlet  | 2 m/s     |
| Entrance      | Velocity-inlet  | 3 m/s     |
| Exhaust       | Pressure-inlet  | 0         |

### Table 3: Boundary conditions of Room 2

| No. of models | Type            | Magnitude |
|---------------|-----------------|-----------|
| Mouth-Cough   | Velocity-inlet  | 2 m/s     |
| Fans          | fan-inlet       | 2 m/s     |
| Entrance      | Velocity-inlet  | 5 m/s     |
| Window        | exhaust-outlet  | NA        |

### 3. Results and discussions

#### 3.1. Results of room 1:

Figure 3: Velocity streamlines of the class entrance of room1

The above figures 3 represents the velocity flow from the entrance of the classroom. The trend seen is that the velocity decreases gradually and then increases as it gets closer to the exhaust.
The cough is generated from mouth and sneeze from nose. Figure 5 shows the flow of velocity caused by the cough. The velocity streamlines from mouth hits the person who is at a distance of 2m. Figure 4 represents the velocity streamlines due to sneeze. It is seen that initially velocity decreases but as it crosses the midsection of the room the velocity increases since it approaches the exhaust.

Figure 5 and 6 shows the velocity streamlines of both cough and sneeze at the same time. It can be observed that the velocity flow pattern of cough and sneeze intersect each other when they are generated from opposite direction. It is also noted that streamlines of sneeze travels more distance with its initial value compared to that of cough before their velocities decreases because the initial velocity of sneeze is greater than that of cough.
Figure 7 and 8 shows the comparison plots between cough and sneeze models. It can be seen that sneeze begins with a higher velocity compared to cough. The velocity flow of both cough and sneeze
decreases up to one-third of the room and then increases. The maximum velocity is observed in the vent region for both the models.

3.2. Results of Room 2:

![Cough streamlines at Location 1 with and without ventilation](image1)

![Cough streamlines at Location 2 with and without ventilation](image2)

The above figures represent cough streamlines in presence and absence of ventilation. Figures 9 represent the velocity streamlines in presence of ventilation while Figure 10 denotes velocity streamlines in absence of ventilation. It can be noticed that the streamlines travel in forward direction when there is no ventilation. In presence of ventilation, the streamlines travel in forward direction for a small distance and travels back in reverse direction due to opposing air velocity from the entrance, thereby approaching the exhaust. Location 2 in Figures 10 is not situated in front of the entrance unlike Location 1 in Figures 9. A similar trend in latter is seen as that of former in absence of ventilation. The velocity streamlines initially travel in forward direction but due to presence of ventilation source by its side (entrance) and the action of fan, it appears to be a swirl pattern.
Figure 11 represents the comparison plots of velocity vs time at location 1 which is in front of entrance. The velocity increases from its initial value when entrance and fans are given some velocity magnitude. It increases suddenly for a particular period due to the air flow from parallel direction (entrance) and then decreases gradually, whereas in case of zero ventilation velocity decreases gradually with time.
Figure 12: Comparison plots of location 2

The above figure represents the comparison of velocity with time in presence and absence of ventilation for location 2 which is not in front of entrance (ventilation source). It can be seen that the velocity decreases for both the cases for a specific time interval after which it increases gradually in presence of ventilation. In the absence of ventilation, the velocity decreases from its initial value as time increases which is similar to Figure 12 without ventilation. Results from room 2 indicate that the virus spreads faster and travel in all directions in presence of ventilation and is tabulated in Table 4.

Table 4: Results of room 2

| S.NO | LOCATION            | WITH VENTILATION       | WITHOUT VENTILATION     |
|------|---------------------|------------------------|-------------------------|
| 1    | Opposite to entrance | Increase by 59.18%     | Decrease by 96%         |
|      |                     | Decrease by 92.5%      |                         |
| 2    | Adjacent to entrance | Decrease by 76%        | Increase by 68.4%       |
|      |                     | Decrease by 92%        |                         |

4. Conclusions

Numerical investigation is carried out to analyze the COVID virus diffusion in a classroom with fan, exhaust, teacher, and students. The streamlines of velocities are reported at various positions in the room. The results from room 1 indicates that all velocity particles of sneeze, cough and entrance first decreased as they move away from the inlets and then increased as they approach the exhaust. It is also seen that streamlines of sneeze travels more distance with its initial velocity value compared to that of cough before their velocities decreases. All the particles move towards exhaust or vent to equalize the inside and outside pressure i.e., to make the gauge pressure zero. It is also seen that velocity from entrance decreases gradually but velocities from nose and mouth decrease suddenly. The intersection of cough and sneeze velocity streamlines indicates that a minimum social distancing of 2m should be followed to prevent the spread of virus. In the presence of ventilation, the velocity increases and then decreases for location 1 (opposite to entrance) whereas, it initially decreases and then increases for location 2 (adjacent to entrance). In absence of ventilation, the velocities at both the locations dropped from their initial values. The velocity increases instantaneously by 59.18 % and then drops by 96% at location 1 when there is ventilation, whereas it decreases by 92.5 % when there is no ventilation. For location 2, velocity decreases by 76% and then increases by 68.4 % in presence of ventilation, whereas it decreases by 92 % from its initial value in absence of ventilation. The total world is adversely
affected due to COVID-19. As time passes people have adapted themselves to live with it. Many research studies are going on to eradicate the virus and to bring life back to normal. This paper concentrates on the velocity flow pattern of cough and sneeze in a confined room and insist the mankind to follow the precautionary measures such as social distancing, wearing masks etc. to fight against this deadly virus.

References

[1] Kao, P. H., & Yang, R. J. (2006). Virus diffusion in isolation rooms. Journal of Hospital Infection, 62(3), 338–345. doi:10.1016/j.jhin.2005.07.019.
[2] Busco, G., Yang, S. R., Seo, J., & Hassan, Y. A. (2020). Sneezing and asymptomatic virus transmission. Physics of Fluids, 32(7), 073309. doi:10.1063/5.0019090
[3] He, Q., Niu, J., Gao, N., Zhu, T., & Wu, J. (2011). CFD study of exhaled droplet transmission between occupants under different ventilation strategies in a typical office room. Building and Environment, 46(2), 397–408. doi:10.1016/j.buildenv.2010.08.003
[4] 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
[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] Balocco, C. Hospital ventilation simulation for the study of potential exposure to contaminants. Build. Simul, 4, 5–20 (2011). https://doi.org/10.1007/s12273-011-0019-6
[7] Peng, S., Chen, Q., & Liu, E. (2020). The role of computational fluid dynamics tools on investigation of pathogen transmission: Prevention and control. Science of The Total Environment, 142090. doi:10.1016/j.scitotenv.2020.142090
[8] 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.
[9] Gao, N., & Niu, J. (2006). Transient CFD simulation of the respiration process and inter-person exposure assessment. Building and Environment, 41(9),1214–1222. doi:10.1016/j.buildenv.2005.05.014
[10] Setti, L., Passarini, F., De Gennaro, G., Barbieri, P., Perrone, M. G., Barelli, M., Miani, A. (2020). Airborne Transmission Route of COVID-19: Why 2 Meters/6 Feet of Inter-Personal Distance Could Not Be Enough. International Journal of Environmental Research and Public Health, 17(8), 2932. doi:10.3390/ijerph17082932
[11] Harish. R, Venkatasubbaiah. K (2013). Mathematical modeling and computation of fire induced turbulent flow in partial enclosures, Applied Mathematical Modelling, vol.37, pp: 9732-9746.https://doi.org/10.1016/j.apm.2013.05.011.
[12] Harish. R, Venkatasubbaiah. K (2013).Transport phenomena of turbulent fire spread through compartment connected to vertical shaft in tall building, Fire Safety Journal, vol.61, pp:160-174.https://doi.org/10.1016/jfiresaf.2013.09.005
[13] Harish. R, Venkatasubbaiah. K (2014) Effects of buoyancy induced roof ventilation systems for smoke removal in tunnel fires, Tunnelling and Underground Space Technology, vol.42, pp:195-205.https://doi.org/10.1142/9789814635165_0024
[14] Harish. R, Venkatasubbaiah. K(2014). Numerical investigation of instability patterns and nonlinear buoyant exchange flow between enclosures by variable density approach, Computers & Fluids,vol.96, pp: 276-287.https://doi.org/10.1016/j.compfluid.2014.03.026
[15] Harish. R, Venkatasubbaiah. K (2015). Large Eddy Simulation of thermal plume behavior in horizontally partitioned dual enclosure, Building Simulation, vol.8(2), pp: 137-148.https://doi.org/10.1007/s12273-014-0198-z
[16] Harish. R, Venkatasubbaiah. K(2015). Numerical study of water spray interaction with fire plume in dual chambers connected to tall shaft, Fire Safety Journal, vol. 74, pp: 1-10.https://doi.org/10.1016/j.firesaf.2015.03.007
[17] Harish. R, Venkatasubbaiah. K(2016). Non-Boussinesq approach for turbulent buoyant flows in enclosure with horizontal vent and forced inlet port, Applied Mathematical Modelling vol. 40, pp: 927-941.https://doi.org/10.1016/j.apm.2015.05.013
[18] Harish. R, Venkatasubbaiah. K(2013). Numerical simulation of turbulent plume spread in ceiling vented enclosure, European Journal of Mechanics-B/Fluids, vol.42, pp:142-158.https://doi.org/10.1016/j.euromechflu.2013.06.001
[19] Harish. R(2018). Effect of heat source aspect ratio on turbulent thermal stratification in a naturally ventilated enclosure, Building and Environment,143, pp:473-486.https://doi.org/10.1016/j.buildenv.2018.07.043
[20] Harish. R(2018). Buoyancy driven turbulent plume induced by protruding heat source in vented enclosure, International Journal of Mechanical Sciences,148, pp:209-222.https://doi.org/10.1016/j.ijmecsci.2018.09.001