Controlling virus droplets diffusion in an isolated room using CFD

N.V.Chandra Shekar, B. Abhinay, C. Raj Pavan, R. Harish*
School of Mechanical Engineering, Vellore Institute of Technology, Chennai, Tamil Nadu-600127, India.
*Corresponding author Email: harish.r@vit.ac.in

Abstract. This paper presents an isolation room model with a patient carrying acute air born disease with an effective ventilation system being functioned. Different models have been simulated with different vent position and the results have been observed. With the unruly rise in the spread of the COVID-19, it is paramount to restrain the virus propagation within the hospital premises. The purpose of the research is to control the virus dissemination caused by sneezing, by altering the position of the ventilation i.e. air inlet and outlet, by using flow across the room to direct it towards the outlet with maintaining a negative pressure. The negative strain helps in confining the air-borne transmission of the deadly virus from spreading across the room and not letting them permeate outside the isolated region. An isolation room model has been studied using computational fluid dynamics, by setting up a discrete phase model by using injection spray modelling to observe the permeation of the virus droplets. The behaviour of these aerosol droplets are studied using simple-semi implicit method for the equation associated with pressure by specifying the droplet size. By altering inlet and outlet locations we are able to minimise the spread of these harmful droplets by using the flow from the air inlet to go against the diffusing droplets. These paper aides well for a sudden isolation room setup anywhere with peruse dimensions of the inlet and outlet height at most optimum position.

1. Introduction
Over the last decade there have been raised concerns on the air borne diseases spread across the globe. Various trials have been carried out by profuse organisations in successfully containing it. An isolation room is one such major aspect, while allowing the sick to develop antibodies and not allowing it to transmit to others. An isolation room of a hospital has exceptional conditions for successful containment of the contagion. Conditions like position of air vents, pressure variations, direction of flow of air inside the room are among required for effective containment. Through years it has been observed how the doctors or the nurses who have regular visit to the infected even with proper safety measures followed end up having got a virus contact. This ends up worse with contact spreading throughout the hospital premises and the worst case is the hospital turning out to be virus spread hotspot. The reason being that they when enter the patients room, they are exposed to air present in the room, the infected coughs/sneezes which in turn the virus which in form of droplets diffuses in the room and makes its presence throughout the room. With the increased awareness of the consistency of indoor air and its association with ACMV (air-conditioning and mechanical ventilation) system with proper design. The knowledge and need for the air change per hour become necessary and a major improvement in containing the air-borne transmissions. Then with the advent of the isolation room major tests were conducted resulting fruitful omission of the spread of the transmission was achieved. By directing the air flow across the room the spread can be swayed and ensure safety. Negative pressure addition plays a key role in ensuring safety environment. The room is generally given negative strain in order to trap the air flowing out of the room via doors or windows. The negative pressure can be created using vents and with the help of the air flow we can prevent the spread of the contagion. Negative pressure prevents movement of air outside the isolated environment. Computational fluid dynamics (CFD) is a field of science based on the solving of numerical problems, most commonly incompressible, viscous flow, including multi component gas mixtures. It is very powerful and authentic tool for investigating the variable airflow patterns regarding the ventilation across its inlet and outlet including variable flow patterns and behaviours. It is considered highly cost effective, explicit way to do analysis. To solve the equations we need some boundary conditions...
which are vital no-slip boundary condition, outlet, inlet, periodic boundary condition. This paper evaluates the air flow patterns through various setups of ventilation arrangements, finding out the appropriate ventilation arrangement for minimizing of virus droplet diffusion. Using computational fluid dynamics (CFD), we observe in detail where an infected person coughs in an isolated room and how it will diffuse. In our simulations, the minimization of virus droplets is achieved by using infection spray modelling.

To our knowledge, the studying of virus diffusion has never done through infection spray modelling. Many researchers investigated the similar issue and literature stated in their respective papers that are relevant to this study has been mentioned. H.Qian and Y.Li [1] explained that the existing ventilation system design based on the Centres for Disease Control and Prevention (CDC) is ineffective in eliminating neither fine nor large respiratory related particles. Thus, they built an isolation chamber with three separate-type exhaust system designs which supplying downwards. The results showed that exhausts at ceiling-level are more systematic at eliminating large amount of air particles situated in an isolated space. Kao and Yang [2] constructed cough model in order to perform arithmetic simulations related to virus droplets diffusion in an isolated room for various types of ventilation arrangements, not only the analysis related to droplet resulted but also the time of dilution of virus propagation for different-type ventilation arrangement. The analysis results proven to be airflow pattern with parallel directional ventilation setup are the most productive method for restraining virus propagation.

Adamset al. [3] have properly designed an Airborne infectious isolation room (AIIR) with anteroom-equipped at varying differential pressures under the conditions of chosen no supplier movement and also with higher supplier movement for studying containment effectiveness of AIIR. Results shown that containment effectiveness with increasing negative pressure differential has resulted in increased effectiveness and with rising supplier movement the effectiveness decreased. Shelly et al. [4] has designed a negative pressure isolation ward with 30 bed capacity introduced with varying ventilation system. The flow introduced has been characterized throughout the room the pressure difference is observed constantly at each and every location inside the ward. The results showed that the room is slightly becoming neutral or positively pressurized when the safety personnel or other staff enters the room, this design has become effective in increasing the capacity in the hospital.

Bhattacharyya et al. [5] evaluation stated that the efficacy of conditioned air emitted from air-conditioning systems to be combined with aerosol sanitizer was examined in order to enter every section of the isolated space and destroy corona-virus. Contemplating factors influencing aerosol sanitizer transporting structures, such as flow dynamics, temperature and turbulent kinetic energy, analysis was done. The results of the analysis show that high turbulent fields in an isolated space are an effective way to distribute sanitizer to eliminate corona-virus. Cheong [7] examined in an isolated room containing negative pressure the patterns of airflow and pollutant diffusion by using CFD modelling based on the implementation of three ventilation systems. The results show that outlet at low level location have better pollutant removal efficiency than at the ceiling level outlet. The study also shows that ventilation techniques and arrangement of furniture have a significant influence on the patterns of airflow and pollutant scattering in an isolated room.

Julian et al. [8] explained the impact of air flow with different types of doors that are used in negative pressure isolated rooms. Single hinged, double hinged, single slide and double slide doors where examined and observed that the double hinged doors while opened have high risk factors of getting the leakage of airflow into or out of the space and sliding door have better efficiency in maintain the required airflow rate. Tung et al. [9] worked on the model of mock-up isolated room with different ventilation arrangement rates and also with pressure differentials of negative values for implementing better ventilation efficiency. Quantitative analysis of ventilation effect is based on two types of indexes namely EI and QL. The numerous experimental [10-13] and analytical models [14-16] developed to estimate the mass flow rates of bidirectional air exchanges in open enclosures [17-19] and rooms. Results obtained that a ventilation system arrangement with a negative pressure differential of value -15.0 Pa in an isolated chamber gives the better possible ventilation efficacy to extract the contaminants out of the isolated system.
Table 1. Different parameters with respect to other researchers.

| Researcher name | Boundary conditions and other parameters |
|-----------------|---------------------------------------------|
| H.Qian and Y.Li [1] | Supply ventilation rate = 12ACH  
Exhaust rate = 13.2ACH  
Emission particles air flow rate = 6.1/min  
Exhale temperature rates = 32 ± 5℃  
Mode particle size = 80 µm |
| Kao and Yang [2] | Cough model conditions = Time ≤ 0.5s, Velocity=8m/s  
and 0.5s < Time ≤ 1s, Velocity= 10e^{-7} m/s  
Constant inlet velocity flow = 0.4m/s  
Outlet pressure = -8Pa  
Room differential pressures = 2.5 to 30Pa |
| Bhattacharyya et al. [5] | Inlet velocity at ceiling = 3.91m/s  
Inlet temperature at ceiling = 24℃  
Mass flow rate = 29.97kg/s  
Inlet velocity of sanitizing machine = 1.5m/s  
Inlet temperature of sanitizing machine = 30℃  
Mass flow rate of sanitizing machine = 1.854kg/s  
Isolation room measurements = 9.144m x 6.096m x 3.658m |
| Cheong [7] | Temperature at air supplied =18℃  
Total air exchange rate = 29.9ACH  
Temperature of all other walls =24℃  
Emission source sphere diameter = 38mm  
Emission species flow rate from source = 0.631/min  
Mass flow rate at extract = 62.6 E^{-6} kg/s  
Isolation room measurements = 3.35 m x 4.8m x 2.5m  
Air-gap underneath the door = 0.9m x 0.012m |
| Tung et al. [9] | Room ventilation supply = 24ACH  
Volume flow rate of air ranges as 85 to 3400m³/hr  
Room differential pressures = 0 to 62.5Pa  
Isolation room measurements = 2.4m x 2.4m x 2.4m |

2. Modelling
A full-scale isolation room present in most of the hospitals is considered. Using Solid works software and ANSYS Space claim we had built a CAD model which includes patient lying on a bed and considering a room with 9ft x 9ft x 8ft. To make sure the effective control of virus containment in an isolated room, we had considered a room with a rectangular inlet and a squared outlet and no other openings so that the air flow can be kept in a stable state. The inlet is a natural vent where we positioned it in such a way that ventilation air directly sweeps the patient’s head and trunk regions, and moves to the ventilation outlet and the outlet was placed on the wall right above the patient’s head. The inner surface of the patient’s human mouth is differentiated by the inlet-mouth boundary condition, because this surface is assumed as the reference boundary of discrete phase corona virus release.

Table 2. Dimensions of CAD model description.

| S.NO | Part Name            | X Direction Length in m | Y Direction Length in m | Z Direction Length in m |
|------|----------------------|-------------------------|-------------------------|------------------------|
| 1.   | Room                 | 1.97m                   | 3m                      | 3.3m                   |
| 2.   | Rectangular inlet vent | 1.4m                   | 0.25m                   | 0.1m                   |
| 3.   | Square outlet vent   | 0.75m                   | 0.5m                    | 0.1m                   |
| 4.   | Bed                  | 0.96m                   | 1.05m                   | 1.9m                   |
5. Human body

|      |      |      |      |
|------|------|------|------|
|      | 0.59m | 0.07m | 1.4m |

Figure 1. Cad model representing patient with room.

3. Meshing
Meshing has been carried out through Ansys workbench and used Fluent as the solver preference. Geometry clean up has been carried out and checked for free edges, scar lines, duplicate surfaces, small fillets and holes and intersection of parts. The element size has been set to 0.03m and the mesh type is unstructured. We have used edge sizing and face sizing for the human body and patient’s bed to attain the good mesh quality (in terms of skewness and orthogonal quality) and for less computational time. The element number is 458943.

Figure 2. Meshing of the room in ansys workbench.

4. Analysis & methodology
We had performed a pressure-based solver. For particle tracking related to discrete phase over time we had made simulation as a transient. The gravity effect on the fluid is given as 9.81m.s⁻². We had used viscous k-epsilon model with RNG and near wall treatment as standard wall function. The
discrete phase model (DPM) is used, since this model allows us to study a mass of particles discretely in a continuous fluid space. The wet particles of the corona virus secreted from the patient’s mouth are considered as a discrete phase and the air flow transmitted through the elevator ventilation valves is considered as a continuous phase inside the interior of the room. The physical models of discrete particles defined in this simulation include two-way turbulence coupling meaning the two-way interaction between continuous and discrete phase. Stochastic collision is activated which means irregular droplets collide with each other. Coalescence is activated means droplets merge with each other, and breakup is activated for collapse of the droplets. The type of discrete phase behaviour will be time dependent and with a time step of 0.001 s by activating the unsteady particle tracking mode. After activating the discrete phase model, the injection process is defined, which determines the type and quality of discrete particles injected into the model. We had defined injection particles as droplet; thus, water is defined as droplets and water vapour is defined as an evaporating gas species. The injection is performed superficially and through the inner surface of the patient’s mouth. According to the definition of injection, human cough virus particles are physically expelled from the patient’s mouth by water droplets that are evaporating in space. These virus droplets have a temperature of 310 K, a velocity of 31.85 m.s⁻¹, and a mass flow of 0.018 kg.s⁻¹, which are emitted at intervals of 0s to 0.1s. The particle diameter of the virus is not constant during propagation and the rosin-rammler-logarithmic distribution method is considered for the size of the diameters. We had activated the species transport to enable the drop mode. The boundary conditions of the discrete phase model are defined as particles at the boundaries of the patient’s mouth and the inlet and outlet of the airflow with Escape mode, for the particles movement through these boundaries. For the boundary of the human body the Reflect mode is applied for the reflection of particles that collide with this boundary, and at the boundary of the side walls of the elevator cabin, the Trap mode is used, which enables that particles are trapped and accumulate in this boundary. Also the inlet ventilation is such that fresh air flow enters the room as a continuous fluid with a speed equal to 2ms⁻¹ and a temperature equal to 291K. The air flow exits from the outlet to the outside environment at a pressure equivalent to atmospheric pressure. We had considered that the air entering the room has a oxygen mass fraction equal to 0.18. The simulation process is performed with a time step size equal to 0.0025s.

| MODELS | | |
|---|---|---|
| Viscous | | |
| K-epsilon model | K-epsilon | |
| Near-wall treatment | RNG | |
| Species model | Species transport | |
| Number of volumetric species | 3(H₂O,O₂,N₂) | |
| Discrete phase model | on | |
| Interaction | Interaction with continuous phase | |
| Particle treatment | Unsteady particle tracking | |
| Physical models | Two way turbulence coupling | |
| | Stochastic collision | |
| | Coalescence | |
| | breakup | |
| Injection | Active | |
| Injection type | Droplet | |
| Release from surfaces | Inlet-nose | |
| Material | Water liquid | |
| Evaporating species | H₂O | |
| Diameter distribution | Rosin-rommler-logarithmic | |
| Point properties | Temperature – 310K | |
| | Velocity – 31.85ms⁻¹ | |
| | Total flow rate – 0.018 kg.s⁻¹ | |

Table 3. Boundary conditions used in fluent analysis.
| Energy | on |
|--------|----|
| **Boundary conditions** | |
| Floor | Wall |
| Wall motion | Stationary wall |
| Heat flux | 0 W.m\(^{-2}\) |
| Discrete phase conditions | reflect |
| Walls | wall |
| Wall motion | Stationary wall |
| Heat flux | 0 W.m\(^{-2}\) |
| Discrete phase conditions | trap |
| Inlet-Air | Velocity inlet |
| Velocity magnitude | 2 ms\(^{-1}\) |
| Temperature | 291K |
| Discrete phase conditioning | Escape |
| H\(_2\)O mass fraction | 0 |
| O\(_2\) mass fraction | 0.18 |
| Outlet-Air | Pressure outlet |
| Gauge pressure | 0 Pascal |
| Discrete phase conditions | escape |
| Inlet-Nose | Wall |
| Wall motion | |
| Heat flux | 0 W.m\(^{-2}\) |
| Discrete phase condition | escape |
| Human body | Wall |
| Wall motion | |
| Heat flux | 0 W.m\(^{-2}\) |
| Discrete phase condition | reflect |

5. Results and discussion

The stop-convergence criterion was selected to be \(1 \times 10^{-4}\) with independent of the results obtained to the mesh density less than 5% contrast to the reference mesh taken. The analysis has been carried out at constant pressure assuming the room at negative pressure. Using ANSYS Fluent post processing vectors, contours and streamline flow across the room has been observed. It has been observed that the flow from the inlet are directed towards to the outlet placed at back of the patients head and the sneeze droplets from the human body are directed directly towards the outlet without affecting the air present in the room. Results of the room temperature have been observed. Figure 3 indicates the streamline flow patterns inside the isolated room and flow patterns originating from the inlet and human nose. Figure 4 indicates the wall shear distribution and maximum values are seen distributed on the upper and lower walls of the room. The time variation of the local inlet velocity, shows spikes whenever the velocity has abrupt changes. Additionally, these spikes are in phase and have the same sign as the wall shear stress. This means that wall shear stress variation during the transient event can be partially described when sudden changes of mean velocity are observed. Figure 5 and 6 shows the streamline pattern and temperature distribution inside the isolated room and it is observed that the
temperature distribution is higher near the doorways.

**Figure 3.** Stream line flow from inlet and human nose.

**Figure 4.** Wall shear analysis of an isolated room

**Figure 5.** Streamline flow from the inlet directing towards the outlet.
6. Conclusions

This paper investigates an isolation room model with a patient carrying acute air born disease with a effective ventilation system being functioned. It was found that the SST k-ω model gave the minimum error comparing with the wall shear stress simulated from other turbulent models, i.e. STD k-ε, RNG k-ε and Realizable k-ε models. Different models have been simulated with different vent position and the results have been studied. The purpose of the research is to contain the spread of the virus caused by sneezing, by altering the position of the ventilation i.e. air inlet and outlet, by using flow across the room to direct it towards the outlet with maintaining a negative pressure. The present study identified that the negative strain helps in confining the air-borne transmission of the deadly virus from spreading across the room and not letting them permeate outside the isolated region. It was also observed that by altering the position of the inlet and outlet the spread and diffusion of harmful droplets were significantly reduced.

References

1) Kao PH and Yang R J2006 Virus diffusion in isolation roomsJournal of Hospital Infection.62(3) 338-345.

2) Adams NJ, Johnson D L and Lynch R A 2011 The effect of pressure differential and care provider movement on airborne infectious isolation room containment effectivenessAm J Infect Control. 39 91-7.

3) Cheong K W DandPhua S Y2006 Development of ventilation design strategy for effective removal of pollutant in the isolation room of a hospital Building and Environment. 41 1161-70.

4) Shelly L Miller PhD, Nicholas Clements PhD and Steven A Elliott 2017 Implementing a Negative Pressure Isolation Ward for a Surge in Airborne Infectious Patients AJIC Journal.45 P652-659.

5) Yun-Chun Tung, Shih-Cheng Hu, Tung-I Tsai and I-Lang Chang 2009 An experimental study on ventilation efficiency of isolation room Building and Environment.44 271-79.

6) Qian H and Li Y 2010 Removal of exhaled particles by ventilation and deposition in a multibed airborne infection isolation room John Wiley & Sons A/S.4 284-97.

7) Yang-Cheng Shih, Cheng-Chi Chiu and Oscar Wang 2007 Dynamic airflow simulation within an isolation room Building and Environment.42 3194–209.
8) Eames, Shoaib D, Klettner C A and Taban V 2009 Movement of airborne contaminants in a hospital isolation room J. R. Soc. Interface. 6 S757–766.

9) Suvanjana Bhattacharyya, Kunal Dey, Akshoy Ranjan Paul and Ranjib Biswas 2020 Oct A novel CFD analysis to minimize the spread of COVID-19 virus in hospital isolation room Elsevier Public Health Emergency Collection - Chaos Solitons Fractals. 139 110294.

10) 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.

11) 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.

12) 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.

13) 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.

14) 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.

15) 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.

16) 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.

17) 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.

18) 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.

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