CFD analysis of the fluid motion in the isolation rooms

P Tamil Selvan¹, K Abhiram², Ch Anu Raghava³ and V Naveen Sai⁴
¹,²,³,⁴ School of Mechanical Engineering, VIT Chennai

Email: abhi.koppaka@gmail.com

Abstract. COVID-19 is declared as a pandemic by WHO (world health organization) which has led to many deaths all over the world. This study deals with the fluid motion in the isolation rooms with 12 or more ACH (air changes per hour) and maintaining a minimum pressure difference of 2.5 Pascal that can help in reducing the transmission of the virus from affected people. ANSI/ASHRAE guidelines are considered for the analysis. These Isolation rooms help in eradicating the spread of the contaminated particles to the surroundings by creating a pressure less than the atmospheric pressure in in the room. CFD simulations are carried to study the fluid motion of the particles emitted by the patient inside the room. The Analysis is carried out with various human cough velocities of different particle diameters and we observed from the results that the time taken by the particles to reach the exhaust increases with increase in particle diameter, and the flow inside the room increases with increase in human cough velocity.

1. Introduction

In the 21st century, there are many hospitals that use isolation rooms to treat patients affected by various diseases such as SARS, TB and current COVID-19 for eradicating the spread. The virus from an infected person is transmitted to others when the person gets in contact with the other person, since these diseases are Airborne when a person coughs or sneeze those particles move around the environment and make people vulnerable. All these rooms are designed in such a way that those particles don’t move out of the room the and are directly passed out of outlet through filters. CFD plays a major role in the construction of these types of chambers by performing various simulations and observing the results obtained. The room is designed in such a way that the inlet air entering the room reaches directly to the patient and leaves the outlet which is placed behind the patient. The air which enters the isolation room is inhaled by the patient and then when he exhales the air leaves through the outlet present in the room which is then sent through the HEPA filters which filters the harmful particles and the clean air is sent to the atmosphere. In the HEPA filters nearly 96% of the particles exhaled by the patient are trapped. According to WHO guidelines, the minimum Air changes per hour should be 12 and the minimum ventilation rate should be 80 L/s. The room was designed in such a way that it meets the criteria of minimum ventilation rate and the minimum Air changes per hour which are theoretically calculated.

Prasad Mahajan et al. [1] learned the significance and the application of a negative pressure chamber and how CFD was used to mimic movement within the chamber. The analysis was performed in a stable condition and the results were observed near the patient by adjusting the parameters of temperature, pressure, and velocity. Sammy Al-Benna[2] described about numerous experimental and enhanced
analytical models to measure the flow rate of dual air exchange in confined spaces and rooms. Yun-ChunTung et al. [3] indicated by the design of ventilation system on controlling the flow of air containing air pollution and prevent the spread to nearby rooms where the door is separated. r. Weihong Guo et al. Indicated that conventional respiration is significant in terms of energy conservation and described the importance of CFD. Ying-Huang Tsai et al. [5] studied used a sophisticated polymerase response method combined with a filter sample to detect the presence of highly potent coronavirus (SARS) in a single patient's room. R.J. Yang and P.H Kao[6] demonstrated about the diffusion of virus in isolation rooms. M.Idrus Alhamid et al. [7] stated that air quality, which prevents the entry of germs should be under consideration to design a negative pressure chamber. In this situation the air filter playing a major role in the chamber. Jinkyun Cho [8] described the flow of air and the distribution of pollutants tested in the air tested by computerized computational fluid dynamics. S.Y. Phua and K.W.D. Cheong[9] has shown the development of an air-conditioning strategy to effectively remove pollution from the hospital, isolation and environment. T.T. Chow a et al. [10] explained about Transformation of the working theater environment from positive to negative pressure.

2. Experimental Analysis
This study deals with the fluid flow visualization in an isolation room by performing various simulations using Ansys Fluent. Energy equation is enabled as the inlet air entering the room is at a temperature of 22°C and the Gravity is given as 9.81 m/s. K-epsilon Turbulence model is considered for the analysis. Injection material is used as Water-liquid. The Boundary Conditions are given as pressure outlet and velocity inlet. The inlet air velocity is taken as 1m/s and human cough velocities are taken as 10m/s,15m/s,20m/s and 30m/s. The results are observed on an Isosurface which is 0.5m above the ground. Discrete state is used for the analysis and the simulation is performed with various particle diameters 3µm,4µm,5µm and 6µm.

2.1 Geometry
The room was designed using solid works with a rectangular inlet and circular outlet, and the dimensions of the room are given below in Table 1. The detailed views of the Design are presented in Figures 1 and 2.

| Sl. no. | Name of the part                  | x    | y    | z    |
|---------|----------------------------------|------|------|------|
| 1       | External Area of the Room        | 3.4  | 2.6  | 3.2  |
| 2       | Bed                              | 1.2  | 0.8  | 1.8  |
| 3       | Legs                             | 0.1  | 0.1  | 0.4  |
| 4       | Human Body                       | 0.18 | 1.5  | 0.7  |
| 5       | Rectangular Inlet                | 1.2  | 0.2  | 0.2  |
| 6       | Circular Outlet Diameter - 0.60m | 1.2  | 2.4  | 2.2  |
Figure 1. Top view of the Room.

Figure 2. Isometric view of the Room.
2.2 Meshing
Model was meshed by using ANSYS Workbench. Edge sizing was selected for more precise results at the edges, with element size is given as 0.01 m for the model and Tetrahedral mesh was considered for the simulation. Table 2 describes the elements and number of nodes present in the Meshed model which is in Figure 3.

![Meshed Model](image)

**Figure 3.** Meshed Model.

| Element quality | Skewness | Nodes | Elements |
|-----------------|----------|-------|----------|
| 1               | 0.911899 | 809696| 2300283  |

**Table 2.** Mesh Details.

2.3 Theoretical Calculations

2.3.1 Air changes per hour.

Air changes per hour = \((0.65 \times \text{wind speed} \times \text{smallest opening area}) / \text{volume of room}\)

- Smallest opening area = \(1.2 \times 0.2 = 0.24 \text{ m}^2\)
- Volume of room = \(3.4 \times 2.6 \times 3.2 = 28.288 \text{ m}^3\)
- Wind speed = 1 m/s

The Required ACH (air changes per hour) after calculation was found to be 19

2.3.2 Ventilation rate.

Ventilation rate (l/s) = \(0.65 \times \text{smallest opening area (m}^2\) \times \text{wind speed (m/s)}\)

The Required Ventilation rate after calculation was found to be 156 L/s
3. Results And Discussion

The Following Figures 4, 5, 6, and 7 gives a clear idea about the particle behavior and fluid flow inside the room, from the figures 4 and 5 it is evident that the cough particles are directly moving out of the outlet without much recirculation in the room.

**Figure 4.** Particle Motion inside the room.

**Figure 5.** Particles Moving Towards the Outlet.

The above Figures depicts the particle movement when the patient coughs or sneeze.
Figure 6. Flow inside the room (Top View)

Figure 7. Flow Inside the Room (Side View)

The above Figures 6 and 7 depicts the flow inside the room when the air enters from the inlet
Figure 8. Velocity at 10m/s.

Figure 9. Velocity at 15m/s.
Figure 10. Velocity at 20m/s.

Figure 11. Velocity at 30m/s.
Simulations Performed Using Various Particle Sizes

3µm, 4µm, 5µm and 6µm

Figure 12. At a Pressure differential of 8pa at the outlet.

Figure 13. At a pressure differential of 14pa at the outlet.

The Air flow patterns in the room are observed by considering the streamlines, from Figures 8,9,10 and 11 it is apparent that the airflow inside the room increases with increasing cough velocity. In Figure 12 it was observed that some of the particles remained in the room when there is an increase in particle size. Our study finds that this problem is controlled by increasing the pressure differential. Figure 13 depicts that Simulations are carried out at a pressure difference of 14 pascal at the outlet with different particle sizes of 3µm, 4µm, 5µm, and 6µm which solves the problem that arrived in figure 12. The Time taken by the particles to leave the room was 19sec.
4. Conclusions
The fluid flow in the isolation room is visualized by performing various simulations. It was observed that the flow inside the room increases with increasing cough velocity. The theoretical calculations indicated that the airflow in the room is in good condition and the patient inside the room won’t suffocate. The results from the present study prove that the pressure difference may not be same for every particle size. The above results also suggest that the particle residence time depends on both the particle size and pressure difference. Particle residence time is directly and inversely proportional to particle size and pressure difference respectively.

5. References
[1] Prasad Mahajan, Arun Saco S, R. Dinesh Kumar, and Thundil Karuppa Raj R. 2018. Airflow Simulation of an Isolation room using CFD Technique. International Journal of Pure and Applied Mathematics, Volume 118, Issue 18, 2018, Pages 4261-4269.
[2] Sammy Al-Benna. 2020. Negative pressure rooms and COVID-19, Journal of perioperative practice, Volume 3, Issue 1-2, December 1 2020, Pages 18-23.
[3] Yun-ChunTung, Yang-ChengShih, Shih-ChengHu. Numerical study on the dispersion of airborne contaminants forms an isolation room in the case of door opening, Applied Thermal engineering, Volume 29, Issue 8-9, June 2009, Pages 1544-1551.
[4] Weihong Guo, Xiao Liu and Xu Yuan. Study on natural ventilation design optimization based on CFD simulation for green buildings, Procedia Engineering, Volume 121, 7 October 2015, Pages 573-581.
[5] Ying-Huang Tsai , Gwo-Hwa Wan , Yao-Kuang Wu , Kuo-Chien Tsao. Airborne Severe Acute Respiratory Syndrome Coronavirus Concentrations in a Negative-Pressure Isolation Room, inspection control and hospital epidemiology, Volume 27, Issue 5, May 2006, Pages 523-525.
[6] P.H Kao, R.J. Yang. 2005. Virus diffusion in isolation rooms, Journal of Hospital infection, Volume 62, Issue 3, 1 March 2006, Pages 338-345.
[7] M. Idrus Alhamid, Budihardjo, and Andre Raymond. 2018. Design of the Ventilation System and the Simulation of Air Flow in the Negative Isolation Room Using FloVent 8.2, AIP.
[8] Jinkyun Cho. 2019. Investigation on the contaminant distribution with improved ventilation system in hospital isolation rooms, Applied Thermal Engineering, Volume 148, 5 February 2019, Pages 208-218.
[9] K.W.D. Cheong, S.Y. Phua. 2006. Development of ventilation design strategy for effective removal of pollutants in the isolation room of a hospital, Building and Environment, Volume 41, Issue 9, September 2006, Pages 1161-1170.
[10] T.T. Chow a, A. Kwan b, Z. Lin a , W. Bai a. 2006. Conversion of operating theatre from positive to negative pressure environment, Journal of Hospital infection, volume 64, Issue 4, December 01, 2006, Pages 371-378.