Proceedings of 10th IOE Graduate Conference
Peer Reviewed
ISSN: 2350-8914 (Online), 2350-8906 (Print)
Year: 2021 Month: October Volume: 10

Computational Study of Location of Return Ventilation in Hvac Systems for Negative Pressure Airborne Infection Isolation Rooms

Sunil Khadka [a], Vishwa Prasanna Amatya [b], Ajay Kumar Jha [c], Bijaya Kumar Sedhai [d]

[a, b, c, d] *Department of Mechanical and Aerospace Engineering, Pulchowk Campus, IOE, Tribhuvan University, Nepal*
Corresponding Email: [a] 075msmde019.sunil@pcampus.edu.np

Abstract

Negative pressure all airborne infection isolation rooms are a specialized application of the hospital's HVAC system, designed to isolate patients with air-borne diseases. During the COVID-19 pandemic in Nepal, many makeshift isolation rooms were built with poor ventilation strategies that increased the risk of cross-contamination. The present study investigates the optimum operational conditions of hospital HVAC systems to minimize cross-infection. Design and simulation of negative pressure isolation room with three different exhaust vent configurations are conducted using Computational Fluid Dynamics (ANSYS software). The predicted behaviour and airflow characteristics of the optimized isolation rooms are assessed for compliance with the prevailing ASHRAE standards. Out of the three cases, the best airflow results are obtained when the outlet vent is kept in the near-patient head. Under this model, the inlet air is introduced to the room from cleaning zone to breathing zone and then flow moves downwards to the return grille. Since most of the dirty zone lies below the knee level of medical personnel, there is less probability of medical personnel inhaling the contaminated area. The temperature is computed at 23℃, pressure obtained is -4.5 Pa, relative humidity is 48%, carbon dioxide concentration ppm is 480 ppm and the air change rate is 13.61. All these parameters comply with the established global standards for HVAC systems.

Keywords

Negative pressure room, Isolation Room, CFD, COVID-19, HVAC, HVAC for hospitals, AIIR

1. Introduction

Negative pressure isolation rooms are dedicated hospital rooms commonly used for patients with infectious diseases especially airborne diseases. The isolation rooms incorporate ventilation systems maintained at a negative pressure with respect to the corridor for facilitating dilution of air contaminants as fast as possible for infection control. These isolation rooms are a specialized application of the hospital's Heat, Ventilation, and Air Conditioning (HVAC)system. The HVAC system controls the central variables related to infectious disease transmission: temperature, humidity and airflow pattern, for mitigating the spread of these infections. Increasing ventilation rates inside the isolation rooms reduce the risk of cross-infection of infectious diseases. Hence it is important to design and implement a proper HVAC strategy for negative pressure isolation rooms. During the COVID-19 pandemic, dedicated negative pressure rooms are known as Airborne Infection Isolation Rooms (AIIR) was prioritized by the Center for Disease Control (CDC). This included retrofitting of existing hospital rooms as well as the construction of new isolation rooms to eliminate re-circulation and cross-contamination of air. The project focuses on the design of optimized negative pressure isolation rooms for COVID-19 infection control within the hospital. Different ventilation designs and HVAC strategies are simulated using ANSYS to optimize the airflow pattern for effective infection control in the isolation room. The results must maintain all the thermal comfort and proper air ventilation standards.[1] [2]. The configuration of outlet vents is very important in isolation rooms as different outlets result in different airflow and different results. Lack of proper ventilation designs, re-circulation of air is the major problem for current hospital systems in Nepal and must be strictly avoided. These health care facilities need proper vents configuration to meet optimum health guidelines to maintain proper airflow in the conditions. MERV 14 filters (90% spot filters) is mandatory before the diffuser in supply ducts. [3]

2. Literature Review

Isolation rooms: A concrete feasible plan for the design, characteristics and requirement of the isolation room were proposed that provides necessary foundation for collaborative design decision solving tools using AIDA analysis. [4].

CFD Analysis in Hospital rooms: The Euler method was used to study the spatial distribution and temporal evolution of the concentration of exhaled and sneezing/coughing droplets within the range of 1.0 10.0μm in the office office, that is, mixed ventilation (MV), displacement ventilation (DV) and under-floor air distribution (UFAD) [5]. Relation between the flows induced by the movement of the hinged door and the person passing through the doorway in addition to ventilation systems are obtained from CFD simulations and laboratory experiments.[6]. Memarzadeh and Xu described the relation between movement/exposure of air particles and time by simulating different types of ventilation system to find the best air change per hour.[7].

CFD in Isolation rooms:Using computational fluid dynamics (CFD) method, the effects of a moving person and the opening and closing of a sliding door on room air distribution on parameters like velocity, pressure contaminant fields are studied to find optimum results.[8]. The numerical comparisons of the thermal comfort parameters, characteristics and behavior are checked if they meet the requirements of flow patterns and all the international standards in the reports. [9].Case study of the operating isolation room conditions are checked with their proper factors and results were obtained that were validated with international reports and guidelines.[10]. Using finite volume method. The position of bed, patient and vents should be done so that best optimized HVAC system is provided. The results of this CFD analysis concluded that immuno-suppressed patients should be close to the air supply, and infected patients should be close to the exhaust. [11]. The high turbulent fields can be the most effective method to distribute the disinfect and sanitizer to the entire space in isolation room to kill corona virus. This result is obtained from CFD analysis considering various parameters like temperature, kinetic energy. [12]

International Standards: Air change per hour must be maintained above 12 in isolation rooms. If the air change value is low than 12, then the room is not properly ventilated and there is huge chance of cross-contamination. The temperature should be maintained from 22°C to 27°C, carbon dioxide should be 400 to 1000 ppm and humidity should be between 40% to 60%. [1] [2] The High Efficiency Particulate Air FIlter(HEPA) is mandatory in exhaust ducts with static pressure above 250 Pa.(99.99% efficiency),

Inlet Air diffuser configuration is taken from the research paper [13]with 60 degree vanes angle for better air flow.

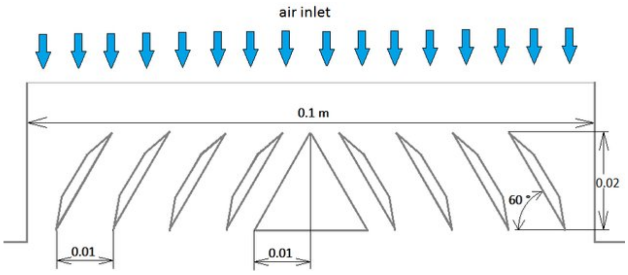

Figure 1: Inlet Diffuser geometry
[13]

3. Research methodology

The flowchart of the research is shown in figure 2.

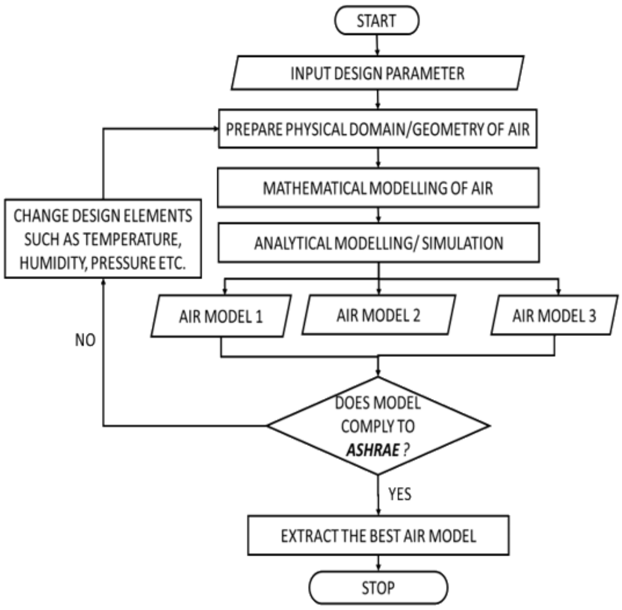

Figure 2: Flowchart process of methodology

3.1 Design Parameters

The room size taken for geometry modelling is 3m*3m*2.5m. The room is properly insulated and made air tight to prevent ex-filtration of air to outside surrounding.The design conditions are taken for Kathmandu, Nepal which is located at 27°4'N 85°21'E and an elevation of 1338m. The average summer temperature varies from 28°C to 31°C.[14]. As such Negative pressure has to be maintained constantly once patients are admitted and system has to be functional continuously until patients are inside the room. Hence, in order to make efficient system, VFD blower motors have to be used which runs in different speed with variable air flow to maintain the desired negative pressure inside the AIIR. The ducts in the supply and return ducts must incorporate HEPA filters static pressure of at least 250 pa minimum (99.99% filtration of 0.3 micron metre). The AHU and blower fan must include the grilles, diffusers and HEPA filter static pressure while calculating the entire external static pressure of the ducts so that the rquired unit capacity and sizing is selected for airflow.

Table 1: Design Parameters

Design Parameters	VALUES(UNIT)	Value
Walls	R(Hr – ft2 – F / Btu)	5.6
Roof	R(Hr – ft2 – F / Btu)	14.48
Light Lux	LUX	300
	LED(lm/watt)	90
Inlet vent	Vent sq. inch	144
	Temp(celsius)	16
	Velocity(m/s)	2
	Humidity(RH)	75%
Patient	Temp(celsius)	38.8
	Heat flux(w/sq.m.)	46.52
	Mass fraction of CO2	0.03861
Doctor	Temperature(celsius)	37
	Heat flux(w/sq.m.)	46.52
Outlet vent	Static Pressure(Pa)	-6

3.2 Physical Domain

Three cases are checked in this thesis, with three different outlet positions.

The size of human patient and doctor is chosen to be of height 1.7m each. The geometry is done in Solidworks. Two lamps of 20W each are chosen and placed on the top. There are three type of cases.

1. AIIR MODEL 1: Outlet 1 feet above ground floor
2. AIIR MODEL 2: Outlet 1 feet below top floor
3. AIIR MODEL 3: Outlet at ceiling

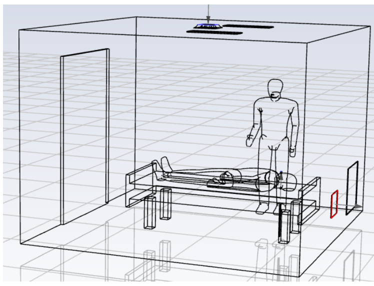

Figure 3: Model AIIR CASE 1

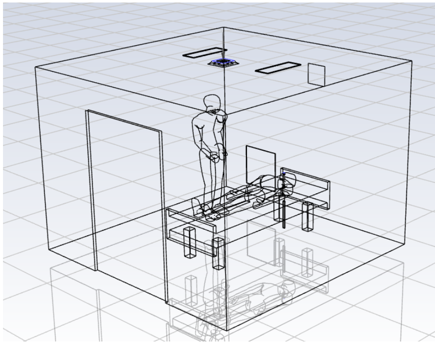

Figure 4: Model AIIR CASE 2

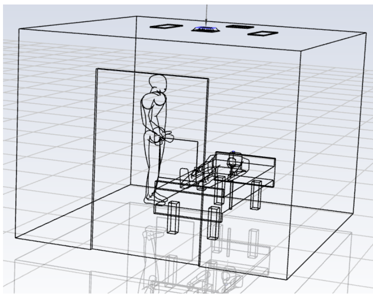

Figure 5: Model AIIR CASE 3

3.3 Mathematical Modelling

Naviers' stroke equation is the main governing equation for a fluid flow and is given by:

$$\rho\frac{\partial u}{\partial t}+\rho(u.\nabla)u = \nabla.[-pI+K]+F\frac{\partial \rho}{\partial t}+\nabla.(\rho u) = 0 \quad (1)$$

For the simulation, The report aims that to numerically simulate the laminar transition flow, the SST k-ε transition model is used. The turbulence model that we used is k-E model whose equation is given as

$$K = (\mu + \mu_T)(\nabla u + (\nabla u)^T) - \frac{2}{3}(\mu + \mu_T)(\nabla . u)I - \frac{2}{3}\rho K I \quad (2)$$

$$\rho \frac{\partial k}{\partial t} + \rho(u \cdot \nabla)k = \nabla \cdot \left[\mu + \frac{\mu_T}{\sigma k}\right] \nabla k + p_k - \rho \varepsilon \quad (3)$$

$$\rho \frac{\partial \varepsilon}{\partial t} + \rho(u \cdot \nabla)\varepsilon = \nabla \cdot \left[(\mu + \frac{\mu_T}{\sigma k})\nabla \varepsilon\right] + c_{\varepsilon 1}\frac{\varepsilon}{k}P_k - c_{\varepsilon 2}\rho \frac{\varepsilon^2}{K} \quad (4)$$

The quantity of air CFM is calculated by

$$\text{CFM} = \frac{\text{Volume of room} \times \text{ACH}}{60} \quad (5)$$

Lighting load is given by

$$\text{Power} = \frac{\text{LUX} \times \text{AREA}}{lm/W} \quad (6)$$

3.4 Analytical Modelling

The mesh is generated of the following study and the simulation is done in ANSYS. The separate mesh for patient and doctor is done first then the individual mesh is done for individual cases.

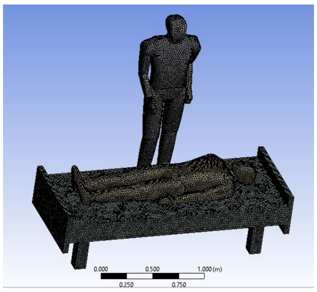

Figure 6: Mesh generation of doctor and patient

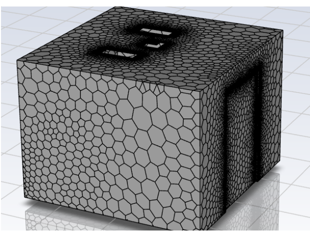

Figure 7: Mesh of Design

4. Results and Discussion

4.1 Temperature profile

The exhaust grille positioning near-patient head helped the supply air to get well distributed in the room. Patient temperature is higher when the person is sick and the design of exhaust location near bed position helped hot air or aerosol to flow short distance in short period of time. Line graph of all AIIR models and ASHRAE Range is plotted in the figure to validate the results with the standards in figure 11.

Figure 8: Temperature profile of case 1

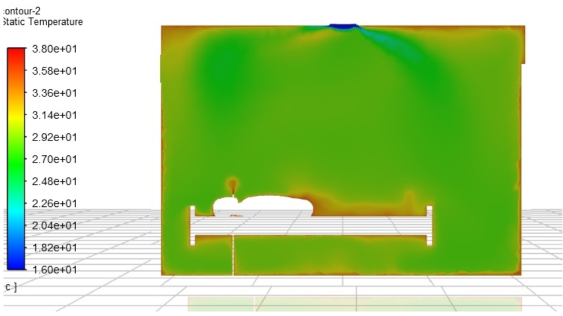

Figure 9: Temperature profile of case 2

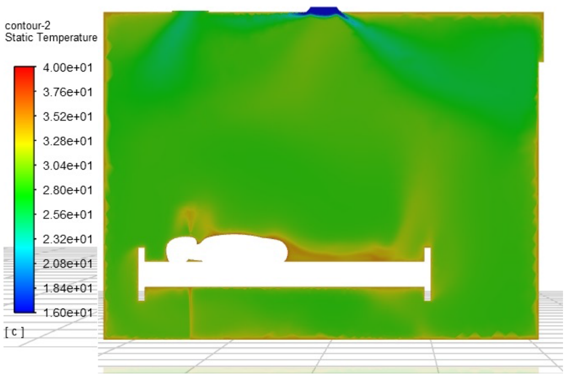

Figure 10: Temperature profile of case 3

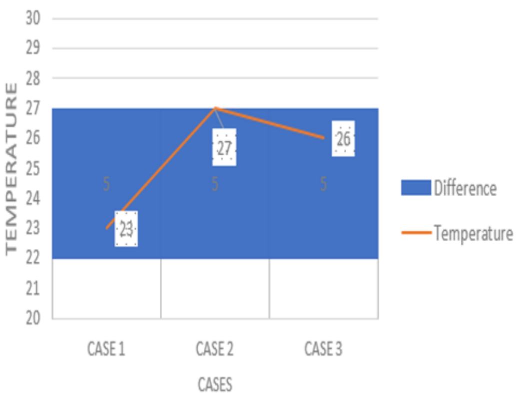

Figure 11: Line graph and range graph Temperature values comparison to ASHRAE standards

4.2 Velocity

The outlet velocity is important in determining air exchange rate and determine the dilution of air inside the room using equation 6. All the models meet the standards of the ACH value of higher than 12 and as seen in figure 10. ACH means how many times the air can be displaced from the space within an hour. The exhaust air flow is higher than the supply and ach more than 12 means the air contaminants inside the room will not circulate outside to the hospital building.

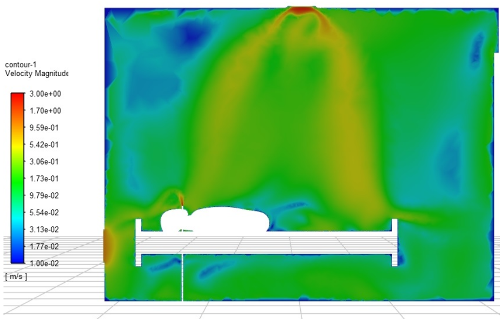

Figure 12: Velocity profile of case 1

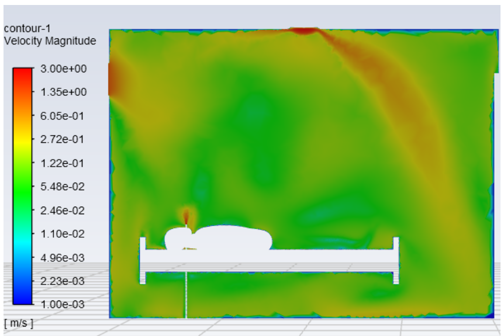

Figure 13: Velocity profile of case 2

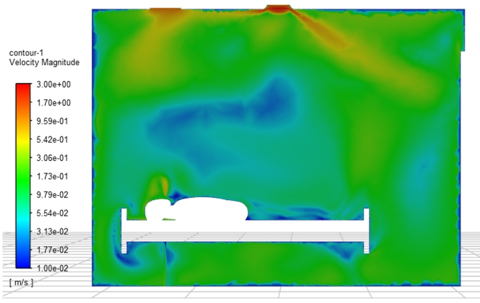

Figure 14: Velocity profile of case 3

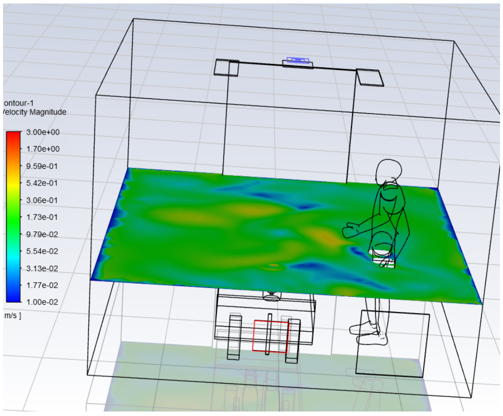

Figure 15: Velocity near patient head of case 1

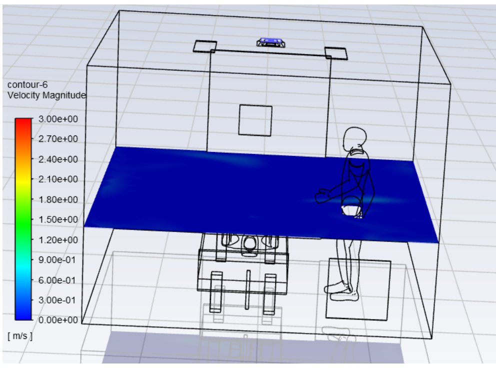

Figure 16: Velocity near patient head of case 2

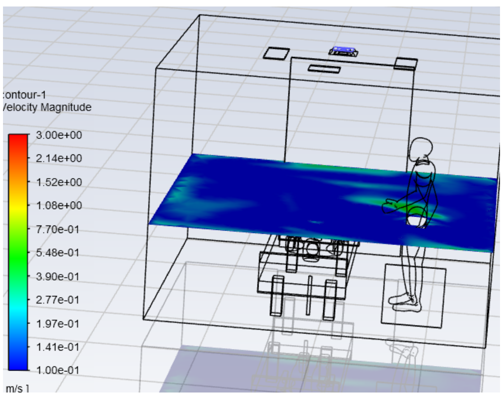

Figure 17: Velocity near patient head of case 3

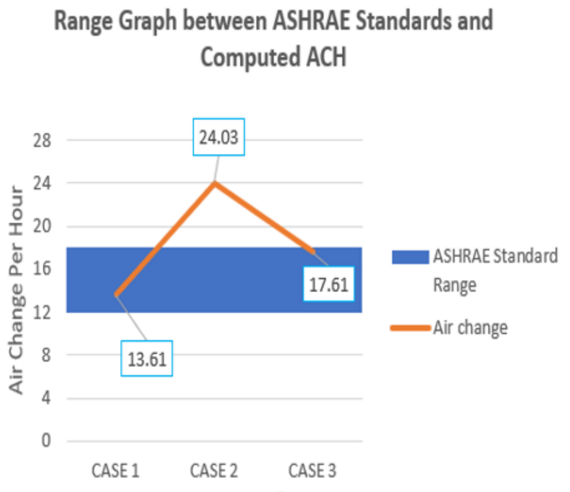

Figure 18: Line graph and range graph ACH values comparison to ASHRAE standards

Table 2: Calculation of ACH values

CASE	Outlet FPM	Outlet area	CFM	Ach
CASE 1	167.3225	1 sq.ft	167.32	13.61
CASE 2	295.275	1 sq.ft	295.27	24.03
CASE 3	216.35	1sq.ft	216.35	17.61

The velocity in case 3 shows stagnant zone near patient as the location of exhaust on the top causes air to be exhaust before reaching the patient and other areas. The velocity near the patient bed also falls under 0.25m/s maintaining the thermal comfort parameter.

4.3 Humidity

All the cases maintain humidity value from 40 to 60 RH percentage as in the figure 19, 20 and 21.

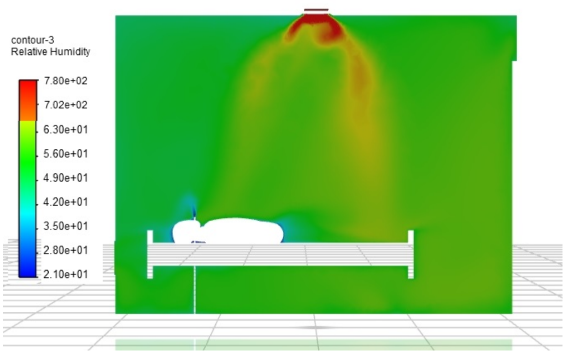

Figure 19: Humidity profile of case 1

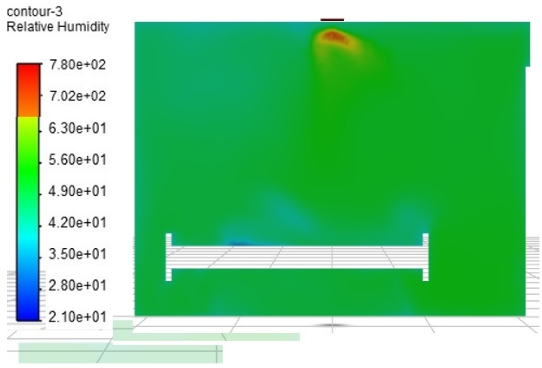

Figure 20: Humidity profile of case 2

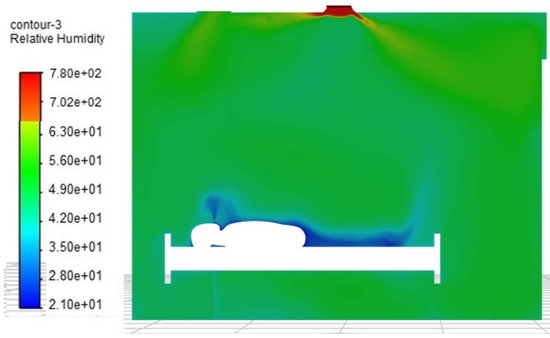

Figure 21: Humidity profile of case 3

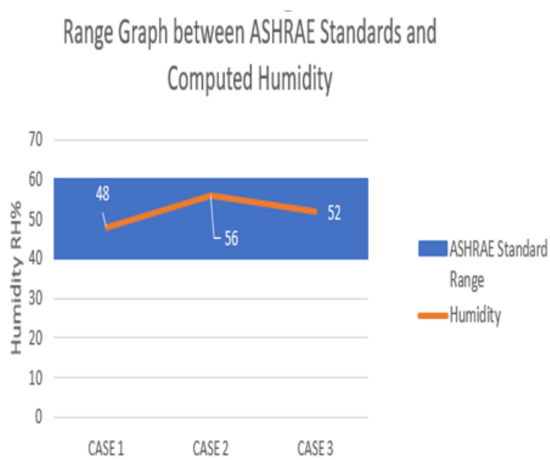

Figure 22: Line graph and range graph Humidity values comparison to ASHRAE standards

4.4 Pressure

The pressure is maintained at least minimum -2.5 Pa with respect to the corridor,[3]. The blower fan must be selected properly including the entire external static pressure of the ducts. If negative pressure is not maintained, it is not safe for medical personnel's to enter inside the isolation room.

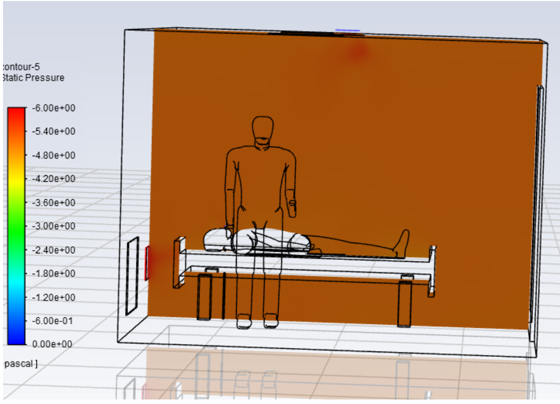

Figure 23: Pressure profile of case 1

Figure 24: Pressure profile of case 2

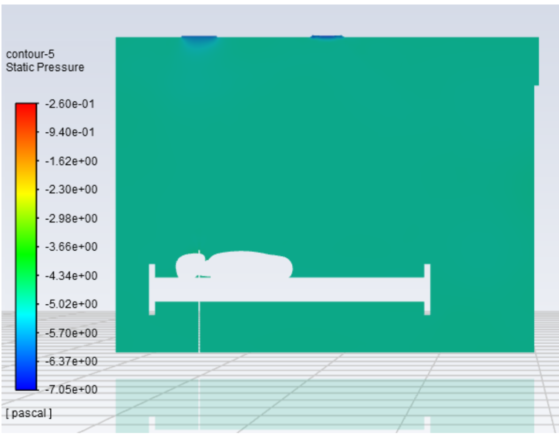

Figure 25: Pressure profile of case 3

4.5 Carbondioxide ppm concentration

The value should be less than 1000ppm in isolation rooms and all cases validate to the standard values.

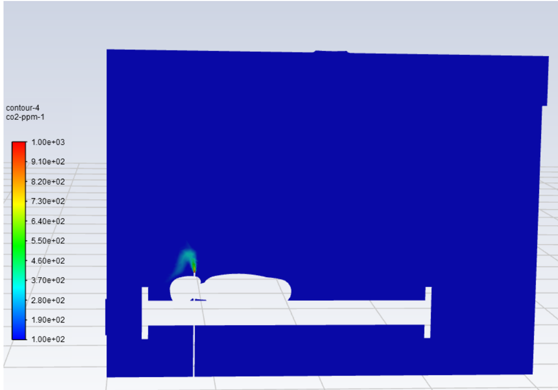

Figure 26: CO_2 PPM profile of case 1

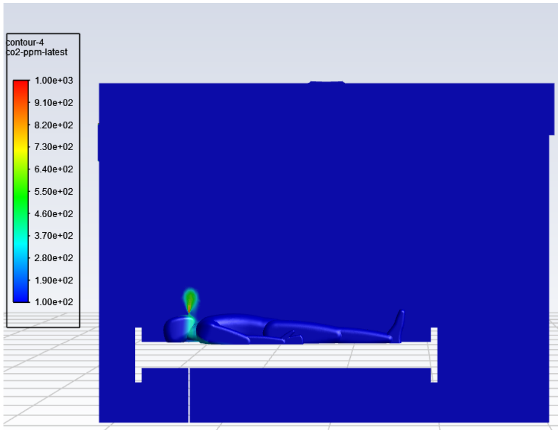

Figure 27: CO_2 PPM profile of case 2

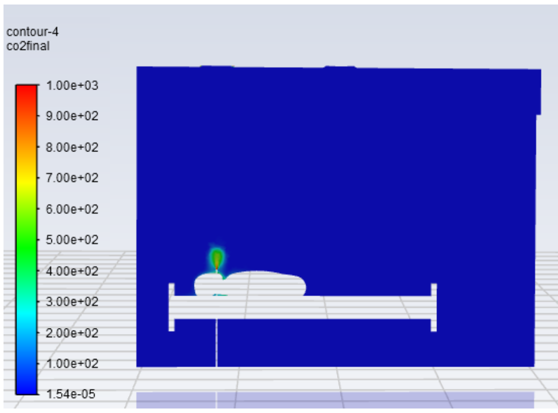

Figure 28: CO_2 PPM profile of case 3

4.6 Velocity vector distribution

The air movement when compared, the air particles move from clean area to the dirty area in model 1 easily and there is proper air distribution.

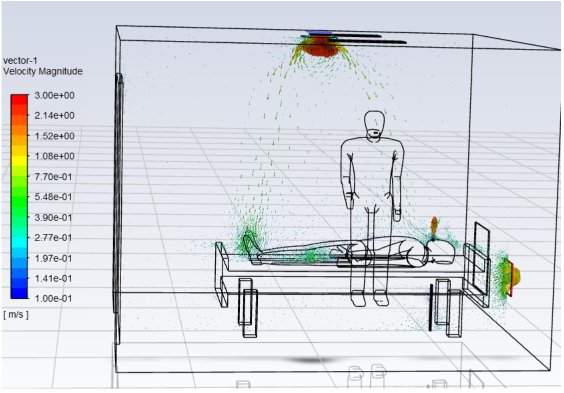

Figure 29: Air Vector Distribution of case 1

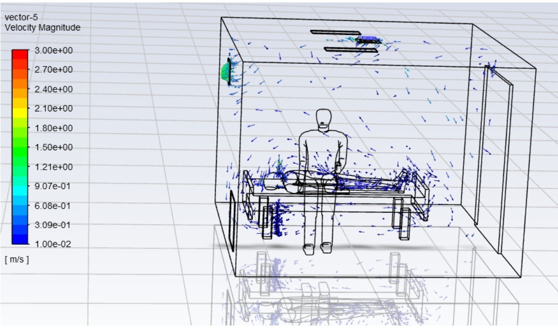

Figure 30: Air Vector Distribution of case 2

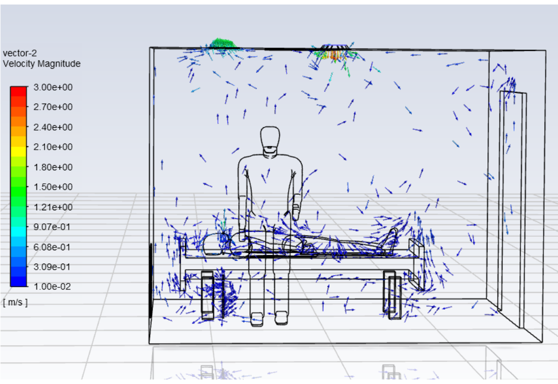

Figure 31: Air Vector Distribution of case 3

4.7 Mesh independence test

The no. of mesh cells are checked to see if the same results are obtained or not to check the validation of the study.

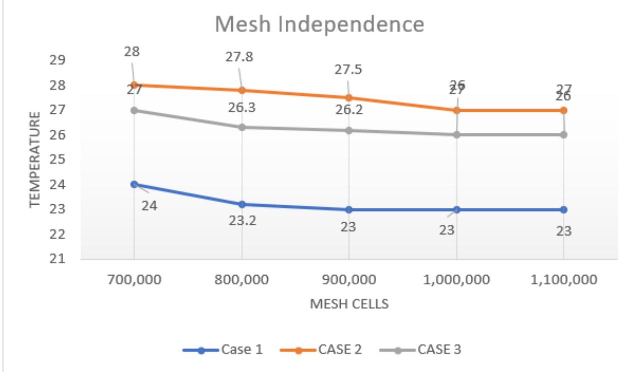

Figure 32: Line graph for mesh independence test in different cases

5. Conclusion

When the outlet is at the top, most of the intake air is exhaust before inlet air can even reach the patient as inlet and outlet are very close to each other. As a result, the stagnant zone is created and the velocity flow is not good. When the outlet is near patient head,the inlet air enters the diffuser to clean zone to breathing area to working area and at the end to the floor to the return terminal. Hence, the downward flow means most of the polluted air or dirty zone is below the knee level of medical personnel and it decreases the probability of inhaling of air by medical personnel. All the models maintain the air exchange rate above 12 i.e. there is the proper dilution of air inside the room. Hence, the air contaminants will not circulate to outer space or hospital areas. The result is also validated with the mesh independence test i.e, when the value of mesh elements are changed, the same results are obtained. Low Air Quality, excessive humidity increases bacterial growth and increase the transmission rate. Hence all these parameters must be maintained in an air borne infection isolation rooms along with proper filtration and irradiation techniques. For constant negative pressure, VFD is mandatory in AHU where as the HEPA filters static pressure should be considered while calculating total external static pressure in the duct system and blower fan capacity. The velocity should not be more near-patient head as it can cause difficulty for the patient. The results conclude that the positioning of outlet vents near-patient head is the better model among the three cases to minimize possible air contamination inside the isolation rooms.

Acknowledgement

The authors like to thank the entire department of Mechanical and Aerospace Engineering, IOE, Pulchowk campus for continuously inspiring me to undertake this research and guide me through this project.

References

[1] Kishor Khankari. Patient room hvac. *ASHRAE Journal*, 58(6):16, 2016.

[2] Lawrence J Schoen. Guidance for building operations during the covid-19 pandemic. *ASHRAE Journal*, 5(3), 2020.

[3] A Bhatia. Hvac design for healthcare facilities. *Continuing Education and Development*.

[4] Tom Behage, Ir RS de Graaf, and X van Ruissen. Identifying the process-based design system of isolation rooms.

[5] Naiping Gao, Jianlei Niu, and Lidia Morawska. Distribution of respiratory droplets in enclosed environments under different air distribution methods. In *Building simulation*, volume 1, pages 326–335. Springer, 2008.

[6] Merethe C Lind, Hannu Koskela, Bård Venås, Anders Welde Vikan, Petri Kalliomäki, and Trond Thorgeir Harsem. Designing simplified airborne infection isolation rooms to reduce infection rate in future pandemics. *ASHRAE Transactions*, 125(2), 2019.

[7] Farhad Memarzadeh and Weiran Xu. Role of air changes per hour (ach) in possible transmission of airborne infections. In *Building Simulation*, volume 5, pages 15–28. Springer, 2012.

[8] Yang-Cheng Shih, Cheng-Chi Chiu, and Oscar Wang. Dynamic airflow simulation within an isolation room. *Building and environment*, 42(9):3194–3209, 2007.

[9] Ahmed Eldegwy, Essam E Khalil, Esmail Bially, and Samy Mourad. Numerical investigations of indoor air quality in infection isolation rooms. In *13th International Energy Conversion Engineering Conference*, page 3820, 2015.

[10] M Idrus Alhamid, Budihardjo, and Andre Raymond. Design of the ventilation system and the simulation of air flow in the negative isolation room using flovent 8.2. In *AIP Conference Proceedings*, volume 1984, page 020016. AIP Publishing LLC, 2018.

[11] Pankaj Saha, PMV Subbarao, and Basant Singh Sikarwar. *Advances in Fluid and Thermal Engineering: Select Proceedings of FLAME 2018*. Springer, 2019.

[12] Suvanjan Bhattacharyya, Kunal Dey, Akshoy Ranjan Paul, and Ranjib Biswas. A novel cfd analysis to minimize the spread of covid-19 virus in hospital isolation room. *Chaos, Solitons & Fractals*, 139:110294, 2020.

[13] Tomas Korinek and Karel Frana. Smoothed particle hydrodynamics simulations of flow in air diffuser. In *AIP Conference Proceedings*, volume 1768, page 020006. AIP Publishing LLC, 2016.

[14] ISHRAE 2019. Nepal weather data. *iSHRAE Journal*, 2019.