CFD INVESTIGATION OF A HOSPITAL'S ISOLATED CLEAN ROOM'S AIR FLOW PATTERN AND PARTICLE TRANSPORT

Favas A.S[1] and B. Bavanish[2]

[1]Research Scholar, Mechanical Engineering, Noorul Islam Centre for Higher Education, Kumaracoil Tamil Nadu, India

[2]Associate Professor, Fire Technology and Safety Engineering, Noorul Islam Centre for Higher Education, Kumaracoil Tamil Nadu, India

ABSTRACT
Globally, Coronavirus spread causes concern and worry among society at large, in particular among healthcare workers as they are at an increased risk for infection. It is an infectious virus that transmits through inhalation or contact with droplets that are produced by people who are infected when they sneeze, cough, or speak. COVID-19 can also be transmitted by airborne means in a confined environment within the immediate environment of the infected person. This study has been conducted to determine whether conditioned air released from air conditioners can be mixed with aerosol sanitizer to reach every corner of the isolation room and kill the COVID-19 virus. This study considers several factors affecting aerosol sanitizer delivery systems such as temperature, turbulent kinetic energy, and flow dynamics that were taken into account during the Computational Fluid Dynamics (CFD) analysis. In this CFD study, the problem is handled with the SST k−ε model which involves four transport equations. From the analysis, it was concluded that high turbulent fields generated inside the isolation room could efficiently distribute sanitizer in the isolation clean room space to reduce or kill the COVID-19 virus.

Keywords: *COVID-19, Airborne, Conditioned air, Isolation room, Turbulent kinetic energy, Clean room, Computational Fluid Dynamics (CFD*

INTRODUCTION

Covid-19 is a virus that causes illnesses ranging from the common cold to more serious illnesses including Middle East respiratory syndrome (MERS)-CoV and severe acute respiratory syndrome (SARS)-CoV (Andrews *et al.*, 2020). Coronavirus is a positive single-strand enveloped, RNA virus ranging from 60 nm to 140 nm in diameter, it is a large family of viruses and it comes under the subfamily of Orthocoronavirinae, it is because of the "crown-like" spikes on their surface(Wu, Chen and Chan, 2019). 2019-nCoV infection, named COVID-19 is a fifth-category notifiable communicable disease that originated in Wuhan at China on December 31, 2019. 98.6 % fever, 69.6 % fatigue, and 59.4 % dry cough are the common symptoms of early clinical causes from Wuhan, China. The spread of COVID-19 disease is very fast because within 2 months (December 31, 2019, to February 17, 2020) 1,772 deaths and 70,635 confirmed cases are reported in China. In this short period, the disease was spread to 25 other countries along with 794 cases and three deaths [3]. The virus was identified and named coronavirus on 7th January 2020. The coronavirus had >95% homology with the bat coronavirus and > 70% similarity with the SARS-CoV,(Singhal *et al.*, 2020). Due to the massive spread of the virus, India and other countries put screening machines in the airport to detect symptomatic people returning from China and other countries and placed them in isolation, and testing them for COVID-19. It was quickly discovered that the virus may be transmitted from asymptomatic to symptomatic persons and even before symptoms appeared. As a result, nations such as India, who evacuated their citizens from Wuhan through special aircraft, isolated everyone who was symptomatic and otherwise for kept them under observation for 14 days after testing. In India, the first case of COVID-19 infection was reported on January 27, 2020, in Kerala. In Kerala, the first infected person is a 20-year-old female who returned to Kerala from Wuhan city (China) on 23rd January 2020 owing to the COVID-19 outbreak situation in China. Due of her travel history from Wuhan, the district quick response team chose to admit her to an isolation room designated for the corona pandemic. 'Isolation is the separation of ill people with transmissible diseases from infected persons to protect non-infected persons, and commonly ensues in hospital settings. Generally, an isolation room could also be equipped with negative pressure to reduce the spreading of the virus via aerosols, but for large droplets like for SARS CoV, there is no need for a negative pressure room, (Freedman, 2020). If early diagnosis is possible before overt viral drop, isolation of patients is very efficient in stopping transmission. Isolation is not an effective strategy to stop the transmission of an influenza pandemic, it is only a controlling strategy, it is because the isolation starts only after to show the clinical symptoms or report the positive for the virus. Similar to community transmission, Severe Acute Respiratory Syndrome Coronavirus 2 (SARS-CoV-2) has also caused healthcare-associated issues in hospitals, leading to concerns regarding the spreading of the disease. SARS-CoV-2 is transmitted not only through direct contact with droplets but also through environmental contamination or airborne transmission in

specific situations, such as during aerosol-generating procedures (AGPs). An isolation room is important to reduce the airborne transmission of SARS-CoV-2 from infected persons to susceptible patients and other persons in hospitals, this room is designed in such a way that it provides appropriate air handling and good ventilation,(Jacob, Yadav and Sikarwar, 2019). An isolation room is built and equipped with all of the necessary amenities to guard against virtually all common routes of infectious microbe transmission, such as contact, droplet, and vector-borne.The designing and implementation of an HVAC system for an isolation room is a serious matter, it requires more care and attention because the most predominant mode of infection communication is airborne transmission. Generally, there are two types of isolation rooms, one is the negative pressure room (Class-N), this isolation room is applicable for those patients who are suffering from infectious diseases like TB and SARS. The aim of admitting infected persons in Class-N isolation rooms is to reduce transmission of disease via airborne transmission. Positive pressure rooms (Class-P) are another form of isolation room. Positive pressure rooms are constructed in specific hospitals to segregate seriously immune-compromised patients, such as those with AIDS.,(Committee and Control, 2007).

To develop a better control strategy for avoiding the further spread of SARS-CoV-2 in a hospital room and for the better utilization of the resources available, it is required to study the trajectory of infectious particles, quality of air, comfort level, and performance of the HVAC system. Numerical techniques such as computational fluid dynamics (CFD) tools can be utilized for this purpose. Along with advanced particle tracking techniques and Computational fluid dynamics (CFD), it will be able to study the air quality inside a room effectively. By fixing all parameters such as temperature and mass flow rate of inlet air, the bed arrangements can be varied for knowing the best arrangement of beds. The effect of sanitized air in the room can be found out by using the particle tracking method.
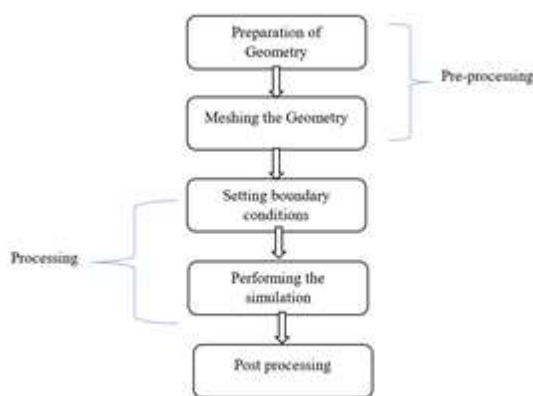
SIMULATION METHODOLOGY

Figure 1: Flow chart of CFD Analysis procedure

Computational fluid dynamics, or CFD, is the computer-based modelling of systems involving fluid movement, heat transport, and related phenomena such as chemical reactions. The equations that control fluid motion can be approximated numerically using CFD. The following steps are required to use CFD to analyse a flow problem. The fluid flow is first described using mathematical equations. A set of partial differential equations is generally used. After that, the equations are discretized to give a numerical equivalent. After that, the domain is split into little grids or elements. Finally,

these equations are solved using the initial and boundary conditions of the specific problem. The technique of solution might be either direct or iterative. In addition, the method's convergence, stability, and accuracy are all controlled by a set of control parameters. There are three main elements in all CFD codes: (1) A pre-processor that inputs the problem geometry, generates the grid, and defines the flow parameter and boundary conditions for the code. (2) A flow solver, which is used to solve the flow's governing equations under the given conditions. A flow solver can utilise one of four methods: I the finite difference technique; (ii) the finite element method; (iii) the finite volume method; and (iv) the spectral approach. (3) A post-processor, which is used to massage the data and show the results in a graphical and easy-to-read format. Figure 1 shows the basic steps of a CFD study.

In this work, the isolation room of a hospital was taken as the physical domain. The isolation room in that hospital is categorized as two, namely unidirectional flow room (Split HVAC room) and multidirectional flow room (Centralised HVAC room). The unidirectional flow room is specially created as an isolation room to handle the pandemic situation. In this room, only one patient is occupied. The multidirectional flow room was the actual isolation room in that hospital. The room is designed for placing 8 beds but due to the pandemic situation and the nature of the disease, only 4 beads are placed in this isolation room. Figure 2 shows the dimensions of the CAD model, which is taken originally from the hospital isolation room. Patient bed, Sanitising machine, air inlet and air outlet (Ventilation), etc are the main component considered in the model. The selected isolation room has 9144 mm length, 6096 mm width, and 3658 mm height. In this CAD model, the patient bed has 2120 mm length, 970 mm width, and 485 mm height. The beds are placed at a distance of 1354 mm. The sanitizer machine used in the room has 250 mm length, 250 mm width, and 550 mm height. In the selected room the HVAC inlet vent has 1066 mm length and 457 mm width. The went is located at the center of the room at a distance of 3658 mm from the floor level. The isolation room is a negative pressure-maintained room, there is only one outlet vent is placed near the bottom of the sidewall. The length and width of the exhaust vent were 1066 mm and 457 mm respectively. The vent is placed at a distance of 450 mm from floor level. According to (Bhattacharyya *et al.*, 2020) the beds are arranged linearly along the length of the room and the opposite side of the exhaust ventilation. However, the proposed model can be meritoriously used for any other combination of physical geometry and bed allocation in the isolation room.
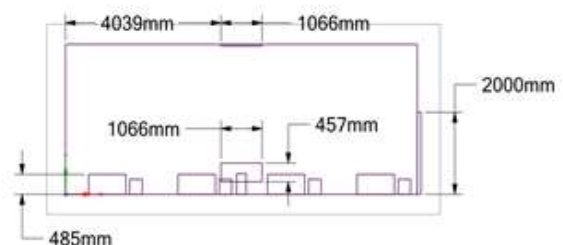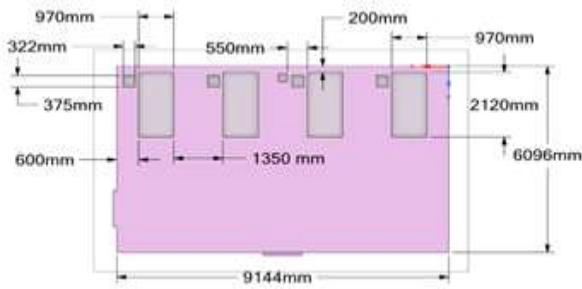
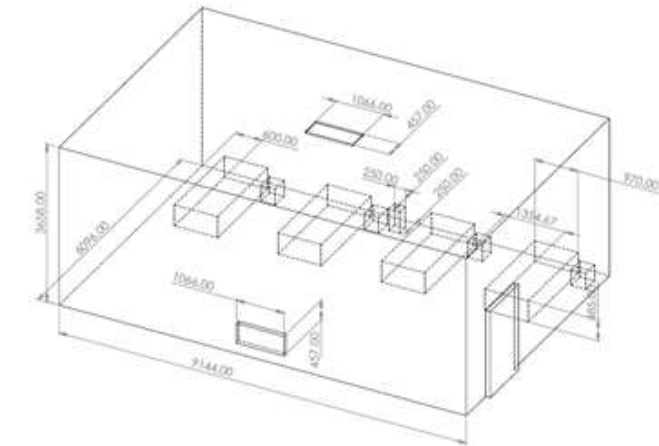**Figure 2:** Schematic diagram of Physical domain

Figure 3: Graphical representation of the mesh dependency study of Element size

After preparing the geometry the next step is geometry meshing. The mesh is created for all selected elements that directly affect the result of the study. Here the mesh was created for various control volumes including 4 beds, sanitizing machine, inlet vent, and exhaust vent.

There are generally 3 different types of meshes are used like fine mesh, medium mesh, and coarse mesh. The selection of mesh type is considered based on the problem and required accuracy. In this work, accuracy is the main factor so here to chose fine mesh. The mesh type depends upon the mesh element size. For obtaining a good result there is a minimum mesh element is required concerning the element size. When the mesh size increases the required mesh element decrease, which reduces the computation time. But it affects the result. When the mesh size decreases the required mesh element for the domain gets increased, which increases the computation time. But to get a good result. The mesh dependency study is one of the effective methods to find the minimum mesh element required without affecting the result. In this problem to chose a fine mesh having a mesh size of 100 mm. Figure 3 shows the graphical representation of the mesh dependency

study having an element size of 100 mm. From the graph, it is clear that after 6 lack the result becomes stable, below that the result is not accurate and that is not taken for the study. Hence in this study to select 675488 elements having the size of 100mm. Here the meshes are generated with Tetrahedral and Hexahedral elements, which is shown in figure 4.
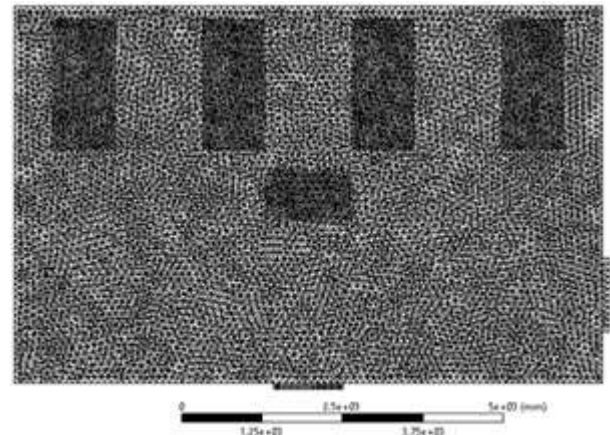

Figure 4: Mesh view of Hospital Isolation room

Boundary Condition plays an important role in any CFD simulation study and the present study like that. The air inlet is placed at the center of the ceiling of the room with a mass flow rate of 0.029 Kg/s, uniformly distributed in the room with an inlet temperature of 21^0C. A pressure outlet with gauge pressure zero is considered at the exit duct of the system. Similarly, the sanitized air coming from the sanitizer machine was 0.000184 Kg/s with an inlet temperature of 30^0C. The applied boundary conditions are shown in Table 1.

Table 1: Boundary Conditions

Boundary	Type	Value	
		Temperature ℃	Mass flow rate Kg/s
Air inlet	Mass flow inlet	22	20
Sanitizer inlet	Mass flow inlet	30	1.85
Outlet	Constant Pressure outlet	Gauge pressure= 0	
Beds	Stationary no slip wall with constant heat flux	90W/m2	
Walls	Adiabatic		

Governing equations play an important role in CFD study. The CFD commonly studies the physics of fluid mechanics and thermal science, for which there are different types of equations are considered like Continuity equations, Navier-Stokes equations, and energy equations, these equations are considered as the main fundamental governing equation used for CFD study(Wah-yen *et al.*, 2017). The above-mentioned equations are commonly used for studies on the field of multiphase flow (Laleh, Svrcek and Monnery, 2011), simple creeping flow(Joseph, 2006), and Couette flow(Taylor, Trans and Lond, 1923), nanofluids motion(Bahiraei, 2016), moving boundaries simulation(Das and Cleary, 2016), and aerodynamic design having complex geometry (Bai and Wang, 2016). The governing equations are developed from the fundamental principles of Newton's Laws and Reynold's transport theorem, which can be generally expressed as in integral form. These governing equations can be also represented in the Eulerian approach for getting precise analysis (Wah-yen *et al.*, 2017). In this study, the problem is handled as air circulation in a close environment containing several obstacles. In such conditions generally, turbulence

models were used. The selection of an appropriate turbulence model for a particular problem was a difficult task because the selection of turbulence model affect the analysis result. Direct Numerical Simulation (DNS), Large Eddy Simulation (LES), and Reynolds- Averaged Naiver –Stokes (RANS) turbulence models are the most commonly used turbulent models (Series and Science, 2018) . The Direct Numerical Simulation (DNS) Naiver-Stokes model is used where the precise result is important, here the model is used without any approximation to compute turbulent flow. When the DNS model is used then to collect 100% accurate data and along with to require large computer capacity. The isolation room is a large enclosed room having many obstacles, in such kinds of problems the use of the DNS model is not practically suitable, there is a high Reynolds number in the flow field which needed high computational capacity and time. Generally, the LES model was used in the macroscopic structure of the turbulent flow. By simulating the turbulent flow using Reynolds- Averaged Naiver –Stokes (RANS) model to use n airflow parameters instead of instantaneous flow parameters. In such kind of problem, both LES and RANS models provide good results, even though the RANS K-Ɛ model is widely used in such kind of research problem because of its simplicity and ease of understanding (Series and Science, 2018). One of the most often used models for capturing the influence of turbulent flow conditions is the k-omega (k−ω) turbulence model. It is part of the Reynolds-averaged Navier-Stokes (RANS) family of turbulence models, which simulates all turbulence effects. It's a model with two equations.That means in addition to the conservation equations, it solves two transport equations (PDEs), which account for the historical effects like convection and diffusion of turbulent energy. Turbulent kinetic energy (k), which defines the energy in turbulence, and particular turbulent dissipation rate (ω), which determines the rate of dissipation per unit of turbulent kinetic energy, are the two conveyed variables. The turbulence scale is another name for it.

RESULT AND DISCUSSION

This work is to study the thermal comfort and indoor air quality in an isolation room of a hospital. The physical study of such kind of problem is very difficult because of its cost, time, and quality of the result. In such a situation CFD study is one of the effective methods to study the characteristics of fluid flow and the particles spreading in a closed area. This particular study is to investigate the optimized position of a sanitizing machine without affecting the thermal comfort in an isolation room providing HVAC ventilation. Four configurations are studied in which the sanitizer machine is placed in locations such as 250mm, 1500mm, 3000mm, 4500mm from the longest side wall near the bedside, which is shown in figure 5.

Figure 6 shows the non-dimensional temperature contour plot in the isolation room when both the sanitizing machine and HVAC system working together. Here the mass flow rate of the HVAC system and the sanitizing machine are constant throughout its working, it is 20 Kg/s and 1.85 Kg/s respectively. Similarly, the air outlet temperature of the HVAC system and the sanitizing machine are constant throughout its working, it is 22^0C and 30^0C respectively.
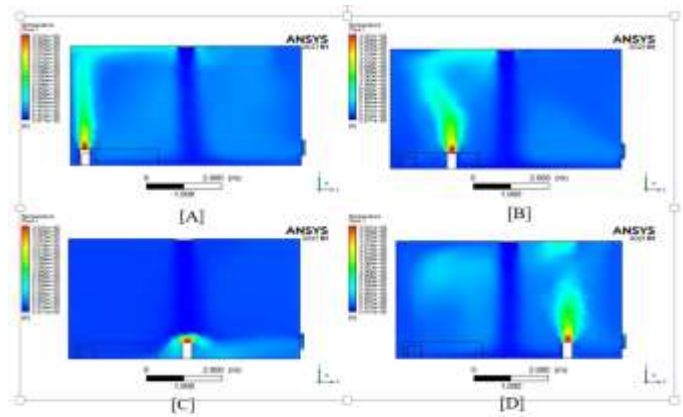

Figure 6: Temperature contour when both the sanitizing machine and HVAC working together (A): Sanitizing machine at 250 mm away from bedside longwall. (B): Sanitizing machine at 1500 mm away from bedside longwall. (C): Sanitizing machine at 3000 mm away from bedside longwall. (D): Sanitizing machine at 4500 mm away from bedside longwall

From figure 6 it is clear that the isolation room maintained a standard air temperature by mixing the sanitizer released from the sanitizing machine at was relatively higher temperature with the cool air coming from the air-conditioning vent. Due to the temperature gradient and velocity, the air coming from the sanitizing machine and HVAC vent become mixed well. An asymmetric pattern appears in the air coming from the air-conditioning vent from the top of the isolation room because of the influence of the velocity and temperature gradient maintained by the sanitizing machine. Figure 7 shows the velocity vector diagram when both the sanitizing machine and HVAC working together at four different positions of sanitizing machine in the isolation room.
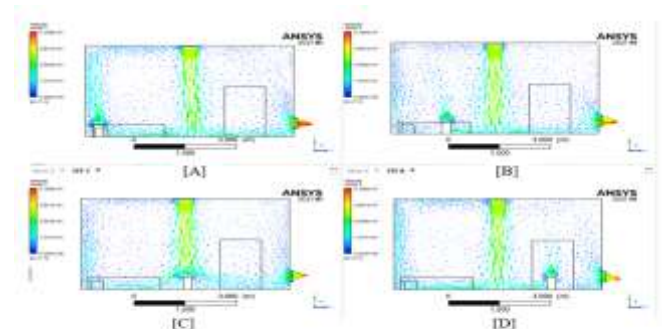

Figure 7: Velocity vector diagram when both the sanitizing machine and HVAC working together (A): Sanitizing machine at 250 mm away from bedside longwall. (B): Sanitizing machine at 1500 mm away from bedside longwall. (C): Sanitizing machine at 3000 mm away from bedside longwall. (D): Sanitizing machine at 4500 mm away from bedside longwall

he velocity vectors are represented in velocity magnitude, that is the fastest fluid flow represents the larger velocity vector. From the velocity vector, it is clear that the flow from the top HVAC vent slows down when it comes in contact with sanitizer flow. When both the flows slow down when it stricks on the walls. Due to the mixing of sanitizer and cool air and bounding walls, there have to form large-scale eddies and flow circulation.

In these four different configurations, configuration D (Sanitizing machine at 4500mm away from bedside longwall) is formed through mixing so that the entire air in the isolation room becomes sanitized without affecting the thermal comfort of the patients.

Figure 8 shows the turbulent kinetic energy contour plot of the mixed air in the isolation room. From these figures, it is clear that there have two vortices are found and high turbulent zones are formed inside the isolation room. The extreme turbulence

generated within the isolation chamber represents full mixing of both airs, ensuring that airborne virus/germs are disinfected using the sanitizer-laden air
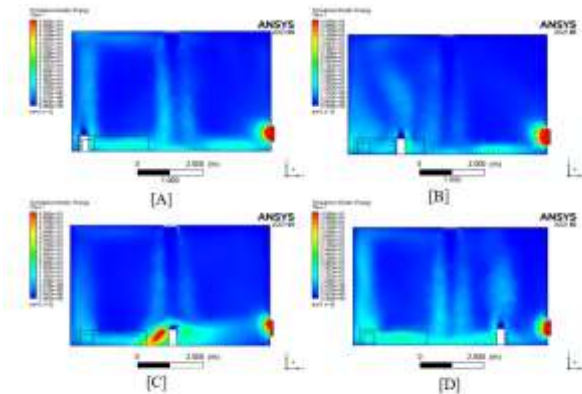

Figure 7: Turbulent kinetic energy when both the sanitizing machine and HVAC working together (A): Sanitizing machine at 250 mm away from bedside longwall, (B): Sanitizing machine at 1500 mm away from bedside longwall, (C): Sanitizing machine at 3000 mm away from bedside longwall, (D): Sanitizing machine at 4500 mm away from bedside longwall

CONCLUSION

Type of patients is one of the major factors that considered the arrangement of an isolation room in a hospital. The isolation rooms for infectious patients and immune-suppressed patients are different from each other. The infectious patient is those who produce infectious micro-organisms through breathing, coughing, and sneezing. A TB or a SARS patient would be a suitable example of such kind of patient. Coronavirus disease (COVID-19) is an infectious disease caused by the SARS-CoV-2 virus. When an infected person coughs, sneezes, speaks, sings, or breathes, the virus spreads in minute liquid particles from their mouth or nose. Larger respiratory droplets to smaller aerosols are among the particles. As a result, a minimum of 1-meter distance between the patient bed and the isolation room is maintained.In the isolation room to provide an HVAC system to maintain good thermal comfort to the patients. The disinfection of the circulated air in the isolation room becomes a major challenge. Air sanitization using a sanitizing machine is one of the suitable disinfection methods in an isolation room. But the usage of a sanitizing machine affects the thermal comfort of the room.

This study was performed to understanding the airflow patterns in the isolation room. This study has been conducted to determine whether conditioned air released from air conditioners can be mixed with aerosol sanitizer to reach every corner of the isolation room and kill the COVID-19 virus. This study considers several factors affecting aerosol sanitizer delivery systems such as temperature, turbulent kinetic energy, and flow dynamics that were taken into account during the CFD analysis. In this CFD study, the problem is handled with the SST k−ε model which involves four transport equations. According to the findings, strong turbulence fields created inside the isolation room may be an effective means of dispersing sanitizer in a confined isolation room volume to kill or reduce the COVID-19 virus. For the particular selected isolation room, configuration D (Sanitizing machine at 4500 mm away from bedside longwall) is suitable to get better disinfection without compromising the thermal comfort of the patients.

In the future, the same model will be used to study the isolation room arrangement having any other physical dimensions. This study focuses on the isolation room arrangement for an infectious patient, but in the future, it also is used to study the isolation room arrangement for immune-suppressed patients

REFERENCES

1. Andrews, M. A. *et al.* (2020) 'First confirmed case of COVID-19 infection in India : A case report', 151(5), pp. 490–492. doi: 10.4103/ijmr.IJMR.
2. Bahiraei, M. (2016) 'International Journal of Thermal Sciences Particle migration in nano fl uids : A critical review', 109, pp. 90–113. doi: 10.1016/j.ijthermalsci.2016.05.033.
3. Bai, C. and Wang, W. (2016) 'Review of computational and experimental approaches to analysis of aerodynamic performance in horizontal-axis wind turbines (HAWTs)', *Renewable and Sustainable Energy Reviews*, 63, pp. 506–519. doi: 10.1016/j.rser.2016.05.078.
4. Bhattacharyya, S. *et al.* (2020) 'A novel CFD analysis to minimize the spread of COVID-19 virus in hospital isolation room', pp. 1–18. doi: 10.1016/j.chaos.2020.110294.
5. Committee, V. A. and Control, I. (2007) 'Guidelines for the classification and design of isolation rooms in health care facilities'.
6. Freedman, D. O. (2020) 'Isolation , quarantine , social distancing and community containment : pivotal role for old-style public health measures in the novel coronavirus (2019-nCoV) outbreak', 27(January), pp. 1–6. doi: 10.1093/jtm/taaa020.
7. Jacob, S., Yadav, S. S. and Sikarwar, B. S. (2019) *Design and Simulation of Isolation Room for a Hospital Á Infection Á Isolation Á Ventilation Á Air conditioning Á.* Springer Singapore. doi: 10.1007/978-981-13-6416-7.
8. Joseph, D. D. (2006) 'Potential flow of viscous fluids : Historical notes', 32, pp. 285–310. doi: 10.1016/j.ijmultiphaseflow.2005.09.004.
9. Laleh, A. P., Svrcek, W. Y. and Monnery, W. D. (2011) 'Design and CFD Studies of Multiphase Separators — a Review', 9999, pp. 1–14. doi: 10.1002/cjce.20665.
10. Series, I. O. P. C. and Science, M. (2018) 'A review on selection of turbulence model for CFD analysis of air flow within a cold storage A review on selection of turbulence model for CFD analysis of air flow within a cold storage'. doi: 10.1088/1757-899X/402/1/012145.
11. Singhal, T. *et al.* (2020) 'A Review of Coronavirus Disease-2019 (COVID-19)', 2019(4), pp. 281–286. doi: 10.1007/s12098-020-03263-6.
12. Taylor, G. I., Trans, P. and Lond, R. S. (1923) 'Stability of a Viscous Liquid Contained between Two Rotating Cylinders', pp. 289–343. doi: 10.1098/rsta.1923.0008.
13. Wah-yen, T. *et al.* (2017) 'Governing Equations in Computational Fluid Dynamics : Derivations and A Akademia Baru Progress in Energy and Environment Governing Equations in Computational Fluid Dynamics : Derivations and A Recent Review', (December).
14. Wu, Y., Chen, C. and Chan, Y. (2019) 'The outbreak of COVID-19 : An overview', pp. 217–220. doi: 10.1097/JCMA.0000000000000270>Wu.