ISSN : 2455-9679
Impact Factor : 4.256

IJRTSM

INTERNATIONAL JOURNAL OF RECENT TECHNOLOGY SCIENCE & MANAGEMENT

"A REVIEW ON HOSPITAL PATIENT ISOLATION ROOM FOR EFFECTIVE UTILIZATION OF POSITION OF AIR CONDITIONING SYSTEM"

Irshad Ahmad [1], Neeraj Agrawal [2]

[1] *M. Tech Scholar Department of Mechanical Engineering , IES, College of Technology, Bhopal MP, India*

[2] *Associate Proessor, Department of Mechanical Engineering , IES, College of Technology, Bhopal MP, India*

ABSTRACT

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. In present paper a Computational Fluid Dynamic (CFD) simulation is performed on the overhead air-conditioning system of a classroom. A CFD model has been developed for position of air conditioner in class room. Originally the position of air conditioner is centrally located.

Key Words: *Air conditioning, CFD, boundary condition, thermal comfort, Fluid.*

I. 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. The inlet and outlet are placed in such a way that they are opposite to each other. The flow inside the room is visualized by performing various simulations with different cough velocities considering different cough particle diameters and the time taken by these particles to reach the outlet were found.

ISSN : 2455-9679
Impact Factor : 4.256

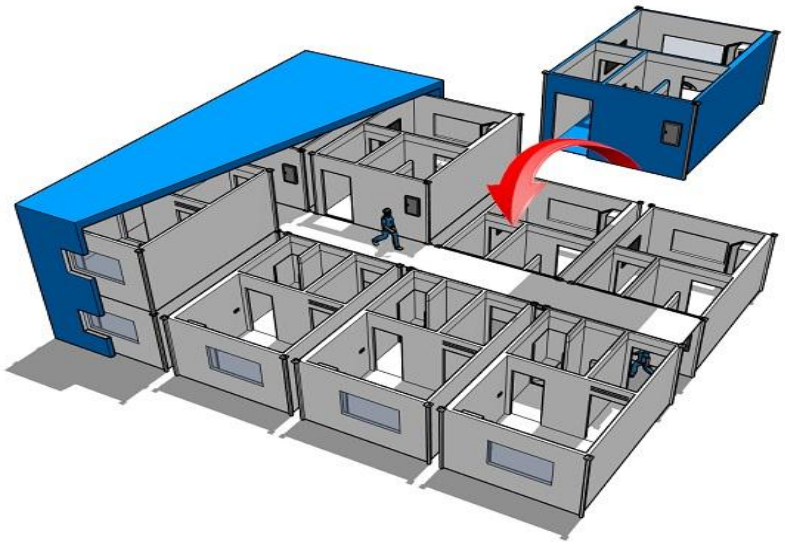

Figure 1.1 Hospital Isolation ward Room

II. LITERATURE REVIEW

Mojtaba Amiraslanpour et al [2020] Impacts of the laminar airflow ventilation system design factors on contaminant removal and thermal comfort condition in an operating room have been investigated by means of Lagrangian-based particle transport using the non-Boussinesq modeling of the buoyancy effects. An operating room including staffs and a patient with realistic human geometries and two surgery lights are included in simulations. The laminar airflow system is placed on the ceiling with a surrounding fixed-height partial wall and the air barrier supply grills. Effects of density change in mixed convection flow regime are included by the non-Boussinesq modeling of the exact air density variation. The predicted mean vote, the age of air, the colony-forming units per cubic meter, the average temperature, the average velocity, the relative humidity, the density distribution, and the positions of particles are calculated to assess the indoor air ventilation quality. A total of 27 simulation cases have been considered to determine the impact of three main design factors including the laminar airflow system area, the supply air, and the air barrier velocities on the performance of the system. It is concluded that for the curtain velocity of 2 m/s, the thermal comfort reduces with increasing the laminar airflow velocity, but for the third staff, the results show that for small laminar airflow areas, the speed of the inlet port should be reduced and for the larger sizes, the inverse of this trend is recommended. Moreover, for all cases the humidity varies within the range of 55%–56%, which agrees well with the suggested standard humidity range between 50% and 60%. It is concluded that the cases of the laminar airflow velocity equal to 0.3 and 0.5 at the curtain velocity of 1 m/s are generally more appropriate than other cases due to less accommodation of particles near the entire body of the patient.

Ashish Mogra et al [2019] The productivity of an individual person affected to a great extent by indoor quality of air and the condition of thermal comfort. Students as well as professors spend almost half of their day in classrooms; hence the distribution of air flow from the air-conditioning systems plays a crucial role in determining whether the students will receive the proper velocity and temperature of air upto the comfortable accepted range. In present paper a Computational Fluid Dynamic (CFD) simulation is performed on the overhead air-conditioning system of a classroom. A CFD model has been developed for position of air conditioner in class room. Originally the position of air conditioner is centrally located. A comparison has been made in between two model placed centrally and proposed diagonally position. It is found the better air flow distribution when air conditioner is placed diagonally.

Shubham Gupta et al [2018]Many researchers have worked on the optimization of different process parameters of UFAD system. To increase the efficiency of UFAD system here in this work it finds out the effect of different shape of supply diffuser. To analyze the effect of different shape it considered three different shapes that is rectangular, circular and triangular. It also finds out the effect of different velocity of air on temperature. Here in this work it also find out the effect of different position of different shape of supply diffuser. After analysis it is found that the circular

ISSN : 2455-9679
Impact Factor : 4.256

perforation at the center of room shows the minimum temperature as compared to the other shapes of diffuser. Here it also finds out the temperature at different height of room. To calculate the temperature at different height here it considered 5 different planes at height 0.5, 1, 1.5, 2 and 2.5 m from the bottom of room.

Mathews, E H et al [2001]Good heating, ventilating and air conditioning (HVAC) control ensures comfort. It is usually also the most cost-effective way to improve energy efficiency of air-conditioned buildings. In this article, the comfort enhancement and energy saving potential with new control strategies are determined for the Human Science Building (HSB) at the University of Pretoria. A new software tool, QUICKcontrol, was used to perform the complex and fully integrated building, HVAC and control simulations. Various control strategies were investigated. These included air-bypass, reset control, setback control, improved start-stop times, economiser control and CO{sup 2} control. The simulation models were firstly verified against measurements to ensure accurate and realistic retrofit simulations. It was then possible to ensure comfort and to predict savings of 60% in HVAC power consumption. This resulted in a simple payback period of 9 months. Preparing input data took about two days, while setting up the simulation model took another day. The typical run time for the fully integrated building, HVAC system and control simulation took approximately 90 s per day on an 'Intel Pentium' 133 MHz personal computer.

III. Problem Formulation

The survey of different previous works we predict the temperature stratification in the most important parameters in order to maintain better comfort conditioning. Poor Efficiency of ventilation can offer reduction in cooling energy utilization of buildings, thus increase risk of infection through air, decrease the level of comfort decreasing comfort in hospital cabins. Thermal discomfort caused by the turbulence nature of air around the benches will be reduced to some extent.

IV. Objective of The Present Work

The objective of the current work is computational experiment work on Air flow.

- By placing one unit above the open space near door, it will possible to eliminate greater turbulence and velocity variations.
- A lower variation of velocity over the sitting area will be observed.
- Variation between maximum temperature and minimum temperature will be comparatively less.

V. Conclusion

The air flow inside the room at two different front sections, at sections close to the window and middle of the room. Air enters the room with a high velocity and after circulating inside the rooms, it also leave the room through the vent with a relatively high velocity; but its exit velocity is lower than the entering velocity. In the sitting elevation level of the room's occupant, the high velocity zones are located in areas close to the rear glass window. The entire area in front of the window always experiences a higher air velocity. Investigated that good Efficiency of ventilation can offer reduction in cooling energy utilization of buildings, thus reducing risk of infection through air, increasing the level of comfort increasing comfort in hospital cabins.

REFERENCES

[1] Holiday, A., Andreou, S., & Hayder(2015). M. Air Flow Visualization in an Office Room, European International journal of science and technology.

[2] Li, Y. (2012). Numerical simulation and analysis for indoor air quality in different ventilation. Health, 4(12), 1352.

ISSN : 2455-9679
Impact Factor : 4.256

[3] Sarkar, J., & Mandal, S. (2008). CFD Modeling and Validation of Temperature and Flow Distribution in Air-Conditioned Space.

[4] K Vaseemul Rahaman, Prof. Shankar Kumar, (2015), CFD Analysis of Air Conditioning equipment of CAD Lab for Enhancing its Performance,International Journal of Recent Development in Engineering and Technology, 1(5) , pp. 68-72.

[5] Sun, Z., & Wang, S. (2010). A CFD-based test method for control of indoor environment and space ventilation. Building and Environment, 45(6), 1441-1447.

[6] Aryal, P., & Leephakpreeda, T. (2015). CFD Analysis on thermal comfort and energy consumption effected by partitions in air-conditioned building. Energy Procedia, 79, 183-188.

[7] Bamodu, O., Xia, L., & Tang, L. (2017). A numerical simulation of air distribution in an office room ventilated by 4-way cassette air-conditioner. Energy Procedia, 105, 2506-2511.

[8] Tian, W., Sevilla, T. A., Li, D., Zuo, W., & Wetter, M. (2018). Fast and self-learning indoor airflow simulation based on in situ adaptive tabulation. Journal of Building Performance Simulation, 11(1), 99-112.

[9] Yang, W., Zhu, X., & Liu, J. (2017). Annual experimental research on convective heat transfer coefficient of exterior surface of building external wall. Energy and Buildings, 155, 207-214.

[10] Wang, H., & Zhai, Z. (2012). Application of coarse-grid computational fluid dynamics on indoor environment modeling: Optimizing the trade-off between grid resolution and simulation accuracy. HVAC&R Research, 18(5), 915-933.

[11] Romano, F., Colombo, L. P., Gaudenzi, M., Joppolo, C. M., & Romano, L. P. (2015). Passive control of microclimate in museum display cases: A lumped parameter model and experimental tests. Journal of Cultural Heritage, 16(4), 413-418.

[12] Popovici, C. G. (2017). HVAC system functionality simulation using ANSYS-Fluent. Energy Procedia, 112, 360-365.