Investigate the effect of change in Geometry and duct position of the patient room using CFD Analysis

OPEN ACCESS

Manuscript ID:

AG-2023-3017

Volume: 2

Issue: 3

Month: September

Year: 2023

ISSN: 2583-7117

Published: 21.09.2023

Citation:

MD Sharib, Pankaj Shrivastava, Dr. B. Suresh "Investigate the effect of change in Geometry and duct position of the patient room using CFD Analysis" International Journal of Innovations In Science Engineering And Management, vol. 2, no. 3, 2023, pp. 121–132.

MD Sharib[1], Pankaj Shrivastava[2], Dr. B. Suresh[3]

[1]*Research Scholar, Department of Mechanical Engineering, Corporate Institute of Science & Technology, Bhopal*
[2]*Assistant Professor, Department of Mechanical Engineering, Corporate Institute of Science & Technology, Bhopal*
[3]*HOD Department of Mechanical Engineering Corporate Institute of Science & Technology Bhopal*

Abstract
The current air quality is one of the issues that must be addressed. The need of oxygen-rich air cannot be overstated. Clean air is becoming a need for human activities. The standard for temperature quality in inpatient rooms is the temperature parameter 22-23℃. This study aims to overcome the distribution of airflow. This type of research uses a descriptive method based on functionality analysis using Computational Fluid Dynamics (CFD). This study uses an HVAC system with variations in geometry, duct position, and cooling time to measure the temperature in the inpatient room. The results showed that if the model of the inpatient room at Bitung Hospital, variations in the location and geometry of the duct. The location of the inlet and outlet diffuser must be on the upper side of the wall, so that it can overcome the temperature of the inpatient room in the hospital. The alternative solution is produced that can increase the maximum thermal comfort for patients at Bitung Hospital. In the result section two geometry get that where thermal comfort is good for patient.

Keyword: HVAC, Inpatient Room, Computational Fluid Dynamics, Heat transfer Coefficient, Indoor Air Quality

Introduction

Ventilation, Heating, and Air Conditioning is an abbreviation for such systems. Enterprise data centers require HVAC systems to be planned for and managed alongside the other data center components like servers, storage, networking, security, and power.[1] These systems regulate the "data center's ambient environment", which includes humidity, temperature, air flow, and air filtration. Almost all pieces of IT gear have certain temperature & humidity requirements.[2] Product specifications and physical planning guidelines often detail these needs. The safety, security, fire, and environmental issues of all of the equipment in the data center must be taken into consideration while designing HVAC system.[3] This is why proper forethought, installation, and upkeep of an HVAC system are essential. In addition, emergency planning has to be included. A data center may, for instance, use HVAC redundancy, stockpile replacement parts, and store portable cooling units as backups.[4][5]

The HVAC system keeps the temperature steady, manages the humidity, and keeps the air clean and healthy. The HVAC acronym stands for the following parts of the system:[6]

Heating

To warm an interior area, the HVAC system employs a heat source, such as a boiler, furnace, or heat pump. A room, floor, or perhaps the whole structure might be the target of the device. Some heating systems employ electricity, heating oil, propane, or natural gas to create heat, while others rely on radiation, convection, or conduction to warm a room.[7] When it comes to the data center HVAC, heat is often not the major issue; nevertheless, in cold areas, heat may be required to preserve exterior equipment or components like chillers.[8]

Ventilation

An efficient HVAC system must have a well thought out ventilation plan. Heating and cooling systems rely on ventilation to keep air moving through a structure, but ventilation is a distinct process.[9] In addition, it brings in outside air to replace the stale air within. Based on the kind of heating and cooling system used, ventilation may also serve to filter the air and keep humidity at a comfortable level.[10] Data centers rely heavily on-air conditioning and ventilation to maintain a stable temperature for their IT hardware. Based on the hardware housed within and the design of the facility, data centers use a wide range of ventilation and cooling methods.[11]

Air conditioning

Air conditioning is often a part of a heating, ventilation, and air conditioning (HVAC) system. Different business spaces may take quite different approaches to cooling. Air conditioning equipment, for instance, might be installed anywhere, even within the structure or on the top.[12] It is possible that water, instead of coolant, will be used for temperature regulation. In addition, it may help ventilation system by filtering the air or regulating the humidity. To keep their IT infrastructure running smoothly, data centers depend significantly on the air conditioning, using a wide range of cooling and ventilation techniques.[13] Although HVAC systems are separate entities, they function as a unified entity to provide the desired environment inside a building. Incorporating cutting-edge technology like deep learning, machine learning, and predictive analytics, the "commercial HVAC systems" have grown more automated and smarter in the recent years. [14]

The importance of HVAC systems in hospitals

The heating, ventilation, and air conditioning systems in hospitals are performing a crucial function by keeping the buildings at a healthy temperature and humidity level and by keeping the air clean and free of germs.[15] Due to these variables, particular consideration must be given in design of the hospital air conditioning systems to a number of criteria that are also relevant in other industries.[16]

More than just comfort for patients and employees

In a healthcare setting, air conditioning does much more than just keep people comfortable. In order to work properly, the medical equipment used in hospitals and other healthcare institutions is very temperature and humidity sensitive.[17] However, the design of such structures is made more complicated by the fact that hospitals must have rooms with quite distinct uses.[18]

It's important to have a well-defined plan for how each area will be used. Some patients in a hospital could be exposed to the "infectious-contagious diseases", necessitating a degree of isolation; similarly, patients with suppressed or the weak immune systems (in intensive care units, neonatal wards, operating theaters, etc.) need to be protected from the plethora of pathogens that thrive in hospitals.[19][20] Because of the high concentration of the pathogens in hospitals, and because most of these chemicals travel in air currents, the air conditioning devices in hospitals is particularly vulnerable to the accumulation of large quantities of the empathogens and can even serve as areas for their cultivation, posing a risk to health of those who use it.[21] All of this may be prevented if the buildings are planned properly to prevent patients, staff, and visitors from being exposed to harmful bacteria and viruses.[22][20]

Zoning in over-pressure & depression is especially important, so consistency when designing zoning of every space to be air-conditioned is crucial. Similarly, designing with the elements which maintain the correct degree of the watertightness, but are at same time accessible and built with the clean materials, is essential.[23] Furthermore, it is crucial to incorporate a sufficient amount of outside air supply, often significantly higher than in the other types of the installations, and a more exhaustive level of the filtration to enable maximum retention of micro-particles that help fungi, bacteria, and viruses that cause diseases settle.[24][25]

Research and methodology

Governing equation

CFD has the potential to solve the momentum, continuity, and energy equations that control fluid dynamics. The "octagonal aquaculture tanks" have a naturally turbulent water flow. For large time-averaged strain rates, the "realizable k turbulence model" is preferable. The "Realizable k turbulence model" was used since flow in the octagonal tanks involves a rotating flow.

Flows in which there is a large unfavorable pressure gradient, the separation of flows, or the secondary flow are all cases in which it excels. Equations that control the system include:

Continuity equation.

$$\frac{\partial p}{\partial t} + \frac{\partial(pu)}{\partial x} + \frac{\partial(pv)}{\partial y} + \frac{\partial(pw)}{\partial z} = 0$$

Momentum equations.

$$\frac{\partial\mu}{\partial t} + div(\mu U) = div(\text{vgradμ}) - \frac{1}{\rho}\frac{\partial P}{\partial x}$$

$$\frac{\partial v}{\partial t} + div(\mu U) = div(\text{vgradv}) - \frac{1}{\rho}\frac{\partial P}{\partial y}$$

$$\frac{\partial\mu}{\partial t} + div(\text{Ï£}U) = div(\text{vgradÏ£}) - \frac{1}{\rho}\frac{\partial P}{\partial z}$$

Energy equation: ("Increase of thermodynamic internal energy of the micro-element body = net heat flow into the micro-element body + work done by the volume force on the micro-element body.").

$$\frac{\partial(\rho h)}{\partial t} + \frac{\partial(\rho\mu h)}{\partial x} + \frac{\partial(\rho v h)}{\partial y} + \frac{\partial(\rho\text{Ï£}h)}{\partial z} = -PdivU + div(\lambda gradT) + \emptyset + S_h$$

Design of patient room

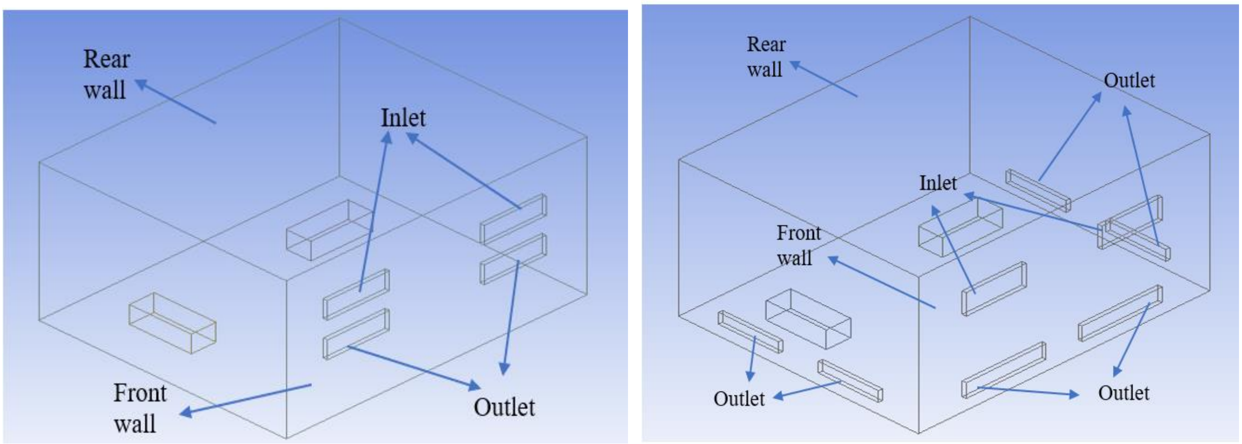
A two-bed, single-patient ward was used to mimic the physical process under examination. The dimensions of the room used for testing were 7.3 meters in length, 6 meters in breadth, and 4 meters in height. As can be seen in the illustration to the right, the length and breadth of both patient beds were 1.5 meters and 0.6 meters. Bed position inside the room is located as 1.2 m away from the front wall for both, and 1m away from left side wall for left bed and also for right bed 1m away from the right-side wall. the design was created in the CATIA software. This parameter is fix for all cases which is considered for this study.

Design of case 1

For case 1 inlet and outlet both are on rear wall. in this case number of inlet and outlet are 2 and both inlets are located at upside and both outlets are located at downside as shown in figure. Dimension of both inlet and outlet are 1.5 m long, and 0.5 m width. Distance of both inlet and both outlet from the roof is 1 m and 2 m respectively. Left side of inlet as well as outlet located at 1 m away from the left side wall. Right side of the inlet and outlet located at 1 m away from the Right-side wall.

Design of case 2

In this case patient room having 2 inlets and 6 outlets. Both inlets are in up side of the rear wall. Rear wall and both side (left and right side) wall is having a 2 outlet each. Length and width of the inlet are 1.5 m and 0.6 m respectively. In this case outlet having a 2 kind of geometry where, 1st type of the outlet having length of 2 m and width of 0.4 m, and 2nd type of the outlet having length and width is 1.5 m and 0.3 m respectively. 1st type of the outlet is located at downside of the rear wall, 0.3 m away from the floor, and 0.6 m away from the both side wall respectively. 2nd type of the outlet is located at both (left and right) side wall 0.3 m away from the floor, 1 m away from the rear and front wall respectively as shown in figure.



(a) **(b)**

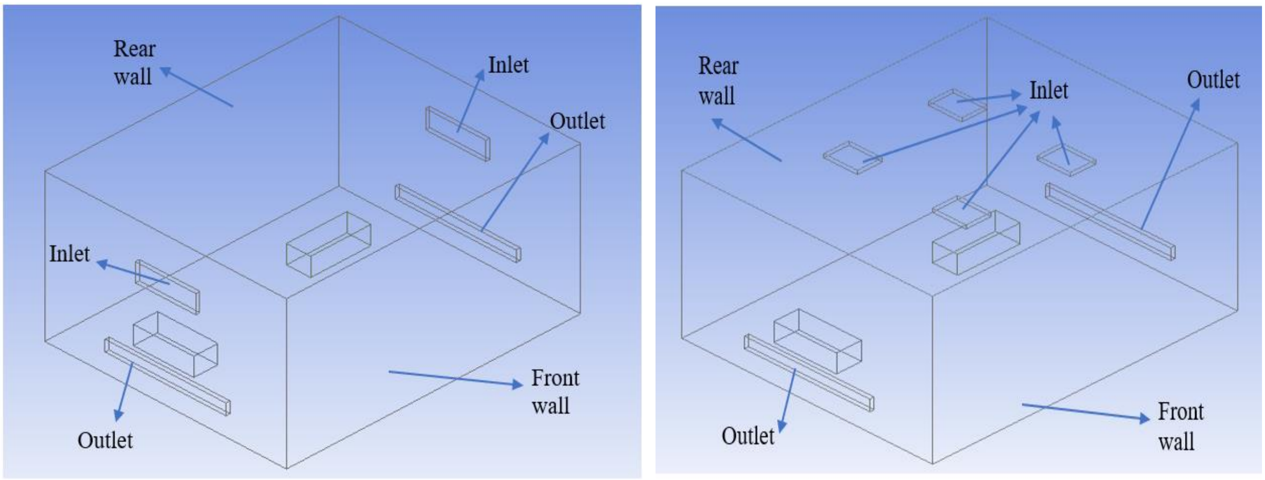
Figure 1 Design of (a) case 1 and (b) case 2

Design of case 3

In this case 2 inlets and 2 outlets are present. 1 inlet and 1 outlet are in each side wall. Inlet having a 1.5m of length, and 0.5 m of width. Located at 1 m down from the roof and equally distance from the both front and rear wall. outlet having 3 m length and 0.3 m width. Located at 0.5 m up from the floor and equally distance from the both front and rear wall. This condition is for both side of the inlet and outlet. Design of the case 3 is shown in the figure below.

Design of case 4

In this case, number of inlet and outlet is 4 and 2 respectively. Each outlet are in each side wall and all inlet are in roof. Length and width of the inlet are 0.8 m and 0.6 m respectively. Outlet having a length of 3 m and width of 0.3 m. Inlet are arrange in 2*2 matrix and located at 2.1 m of equally distance from both the side wall and 1.3 m of equally distance from the both rear and front wall. outlet is located at 0.5 m up from the floor and 1.5 m of equally distance from both rear and front wall. The condition is mention for the outlet is for both side wall.



(c) **(d)**

Figure 2 Design of (c) case 3 and (d) case 4

Meshing

The full-scale test room presented above is simulated in the CFD analysis. Meshing element shape is tetrahedral and quadrilateral shape with structure grid for patient room and bed respectively. Number of nodes and element for all case of meshing is mentioned in table below.

Table 1 Meshing elements and Nodes of all cases

	Element	Nodes
case 1	86205	17371
case 2	86819	17567
case 3	87352	17650
case 4	87544	17706

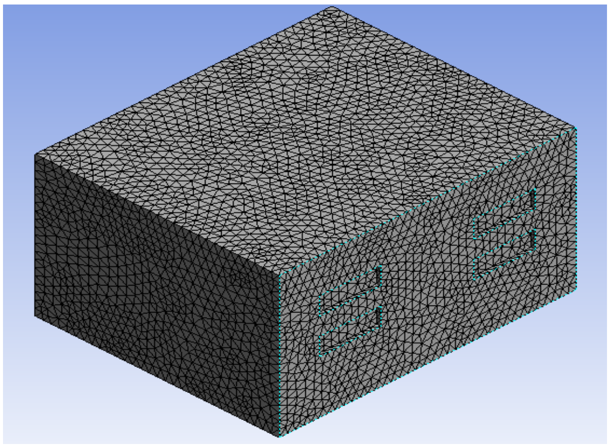


Figure 3 Meshing of Patient room

Boundary condition

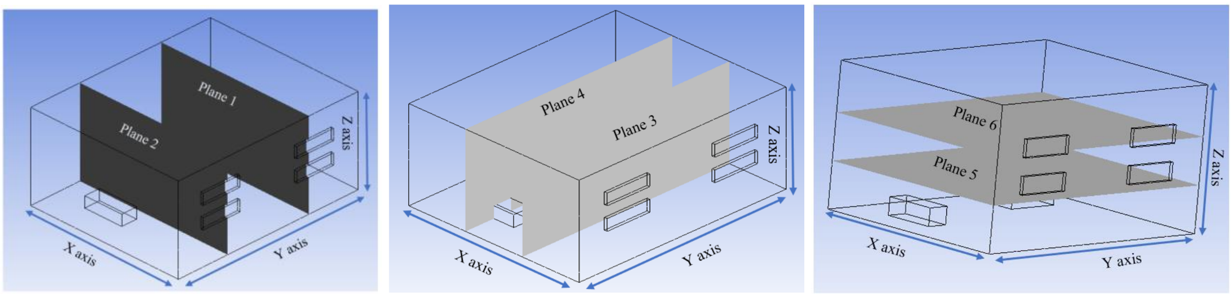
The current air quality is one of the issues that must be addressed. Humans absolutely need air to survive. The standard of temperature quality in inpatient rooms is the temperature parameter 22-23°C. One of the causes of air pollution in inpatient rooms is characterized by the growth of germs. Germs in the air in the inpatient room can come from patients, people who visit hospitals, and health workers. In addition, germs in the air can come from splashes of saliva and dust in the room, as well as caused by room temperature that is not in accordance with air quality standards. The impacts of indoor air pollutants on health include asthma, hypersensitivity to pneumonia, flu and other viral diseases.

The simulation was done under the ANSYS fluent. "Realizable K-epsilon models" with conventional wall functions were utilized because of significance of the turbulence models in the numerical simulation. When using ANSYS Fluent, the numerical convergence as well as residual should not drop below $10^{-3}$for any quantity other than energy, where it should drop below 10^{-6}. SIMPLE scheme for pressure-velocity coupling, second order select for pressure, momentum, and energy, first order for the turbulent kinetic energy, and turbulent dissipation rate. In the initial room temperature is 37 ℃. Air is flow at 0.39 m/s velocity and 20 ℃ temperature. All layers in the room are considered as walls with convection thermal conditions, then use the heat transfer coefficient of 5 w/m^{2}K, a free stream temperature of 40 ℃, and a wall thickness of 0.228 m.

For the comparison of the temperature inside the room generate planes. Total 6 planes are present inside the room which is each 2 planes are along to the length, width, and height. Planes taken in room domain. Location of the planes are mention in the table. Planes are show in the figure below.

Table 2 Planes location

Plane 1	XZ	2 m
Plane 2	XZ	5.3 m
Plane 3	YZ	2 m
Plane 4	YZ	4 m
Plane 5	XY	1.3 m
Plane 6	XY	2.6 m



(a) Plane 1 and Plane 2 **(b) Plane 3 and Plane 4** **(c) Plane 5 and Plane 6**

Figure 4 Planes representation

Result and discussion

Validation

The numerical model is verified by comparing the simulation results with experimental data (Aulia et al., 2023) [26]. CFD investigation is performed in a test room with 2 single bed patient room whose dimensions are 7.3 m long, 6 m width, and 4 m height. The both patient beds had a dimension of 1.5 m long and 0.6 m width. Bed position inside the room is located as 1.2 m away from the front wall for both, and 1m away from left side wall for left bed and also for right bed 1m away from the right-side wall. Number of inlet and outlet are 2 and both inlets are located at upside and both outlets are located at downside at rear wall of the room. Dimension of both inlet and outlet are 1.5 m long, and 0.5 m width. Distance of both inlet and both outlet from the roof is 1 m and 2 m respectively. Left side of the inlet and outlet located at 1 m away from the left side wall. Right side of the inlet and outlet located at 1 m away from the Right-side wall. In the initial room temperature is 37 ℃. Air is flow at 0.39 m/s velocity and 20 ℃ temperature. In figure below compares the present simulation result and (Aulia et al., 2023) [26] of average room temperature at 600 seconds. Excellent agreement between present and experimental CFD findings validates and verifies our model for future study.

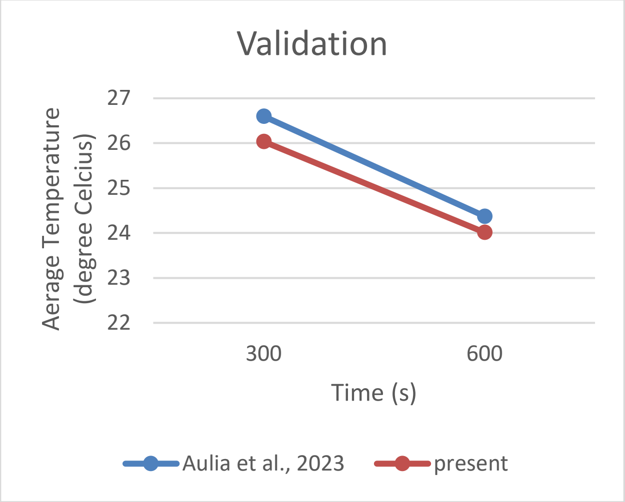
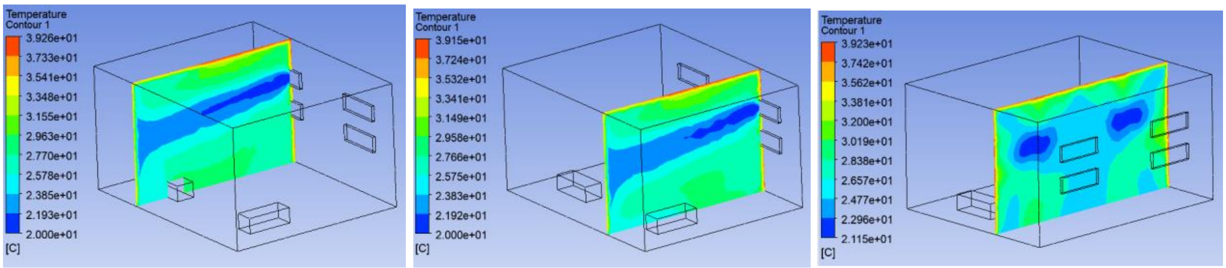


Figure 5 Validation result

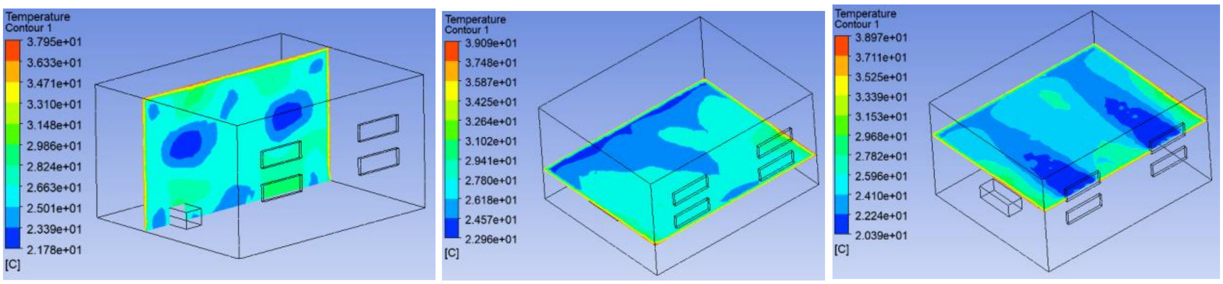
Case 1

Temperature contour

Air entered in the room at 20℃ and as far as reach increases in temperature due to the high room temperature. The figure shows that heat flux emitted from the floor surface disarranges the stratification of the temperature. It is evident from the figure that the highest air temperature in this case belongs to the air flow in vicinity of wall and floor surface (39°C) and that the airflow in vicinity of the outlet vent has the average air temperature (30°C). In the present section, effect of the floor temperature on the fresh air distribution in the breathing zone is evaluated. Temperature distribution are shown in figure in the patient room in different plans.



(a) Plane 1 **(b) Plane 2** **(c) Plane 3**



(d) Plane 4 **(e) Plane 5** **(f) Plane 6**

Figure 6 Temperature contour in all planes of case 1

Velocity vector

In this case at inlet air velocity is 0.39 m/s. it is in decreasing when reaches at opposite wall. when air strike to the front wall forms a vertex to all side. In this scenario fresh air is circulate all around patient. At the outlet velocity is maximum which is 0.5 m/s. in this case inlet and outlet both are on same wall so air circulation is very well at around patient. Velocity vector show in figure below;

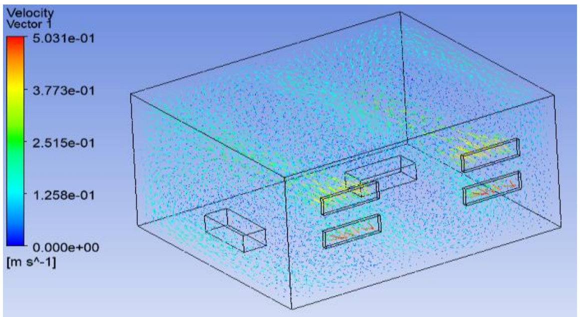
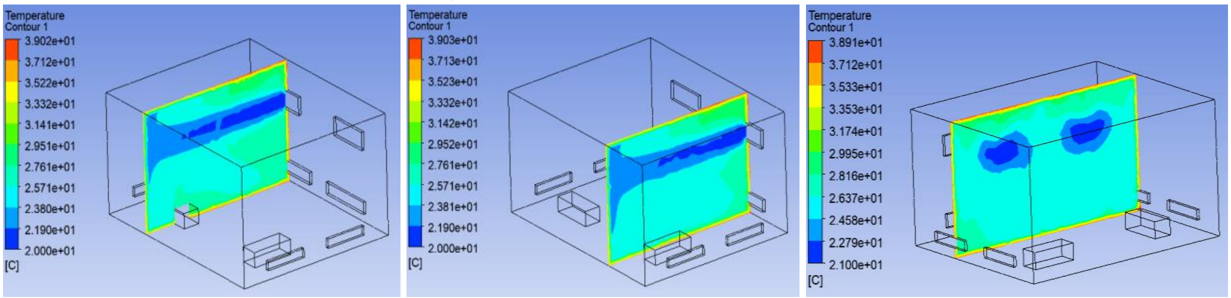


Figure 7 Velocity vector of case 1

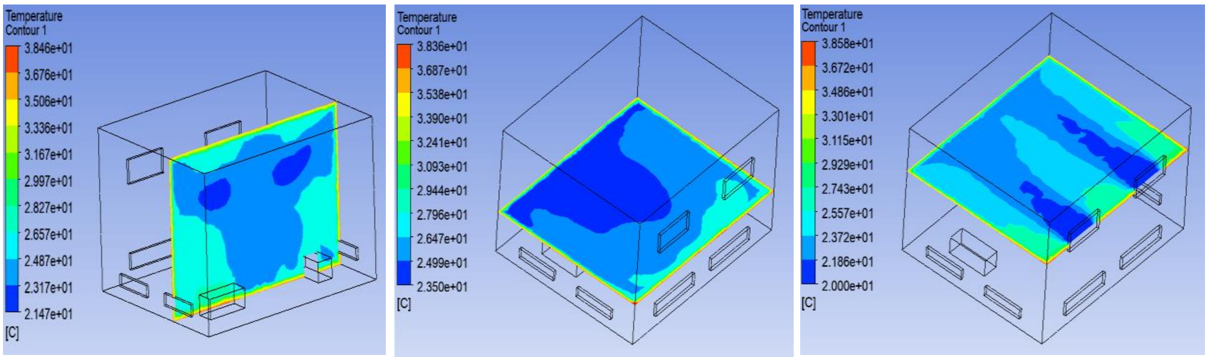
Case 2

Temperature contour

Air entered in the room at 20℃ and as far as reach increases in temperature due to the high room temperature. The figure shows that heat flux emitted from the floor surface disarranges the stratification of the temperature. It is evident from the figure that the highest air temperature in this case belongs to the air flow in vicinity of wall and floor surface (39℃) and that the airflow in vicinity of the outlet vent has the average air temperature (31℃). In the present section, effect of the floor temperature on the fresh air distribution in the breathing zone is evaluated. Temperature distribution are shown in figure in the patient room in different plans. In this case area of the inlet is increases due to that outlet area is also increases. In this scenario 6 outlet are present and arrange in condition of each 2 outlet are in left and right-side wall and rear wall. In that case, average room temperature is decrease from case 1.



(a) Plane 1 **(b) Plane 2** **(c) Plane 3**



(d) Plane 4 **(e) Plane 5** **(f) Plane 6**

Figure 8 Temperature contour of case 2

Velocity vector

In this case at inlet air velocity is 0.39 m/s. it is in decreasing when reaches at opposite wall. when air strike to the front wall forms a vertex to all side. In this scenario fresh air is circulate all around patient. At the inlet velocity is maximum which is 0.39 m/s. In this case outlet are on three walls of the room so air circulation is very well at around patient. In all of the outlet vent average velocity is flow which is 0.25 m/s. Velocity vector show in figure below;

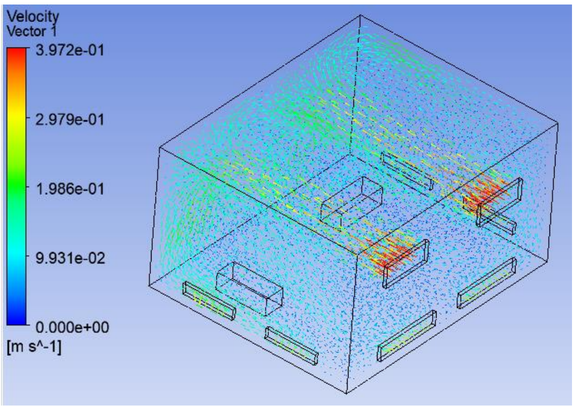
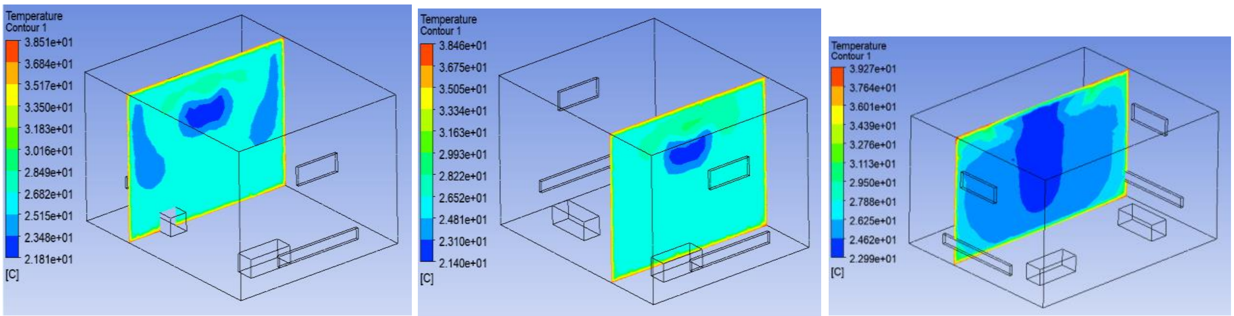


Figure 9 Velocity vector of case 2

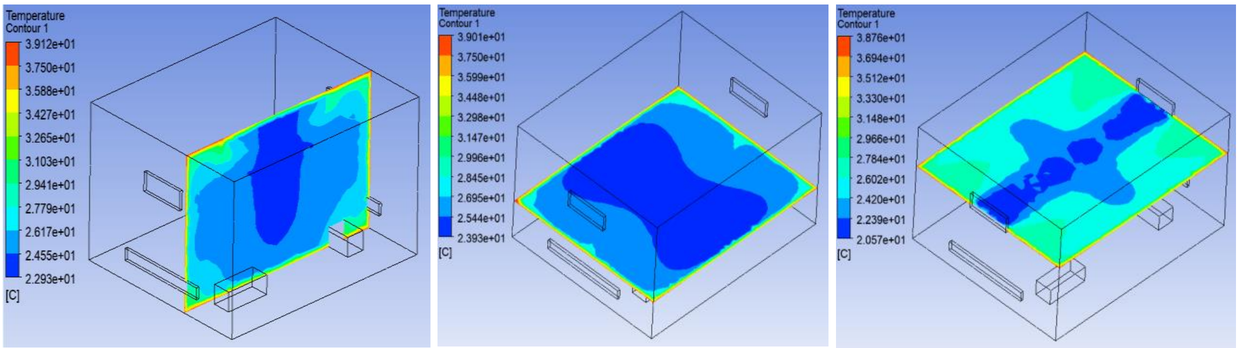
Case 3

Temperature contour

Air entered in the room at 20℃ and as far as reach increases in temperature due to the high room temperature. The figure shows that heat flux emitted from the floor surface disarranges the stratification of the temperature. It is evident from the figure that the highest air temperature in this case belongs to the air flow in vicinity of wall and floor surface (39 − 38℃) and that the airflow in vicinity of the outlet vent has the average air temperature (31℃). In the present section, effect of the floor temperature on the fresh air distribution in the breathing zone is evaluated. Temperature distribution are shown in figure in the patient room in different plans. In this case area of the inlet is same but outlet area is increases. Change the location of the inlet and outlet do investigate the average room temperature. In that case, average room temperature is decrease from case 1. In this case average temperature at surrounded at patient is 26 − 24 ℃.



(a) Plane 1 **(b) Plane 2** **(c) Plane 3**



(d) Plane 4 **(e) Plane 5** **(f) Plane 6**

Figure 10 Temperature contour of the case 3

Velocity vector

In this case at inlet air velocity is 0.39 m/s. it is in decreasing when spread in the room. When air strike with each other forms a vertex to all side. In this scenario fresh air is circulate all around patient. At the inlet velocity is maximum which is 0.39 m/s. In this case outlet are near to the patient bed so, air circulation is very well at around patient. There is area of the inlet and outlet are same so that the outlet and inlet velocity are approx. equal to 0.39 m/s. Velocity vector show in figure below;

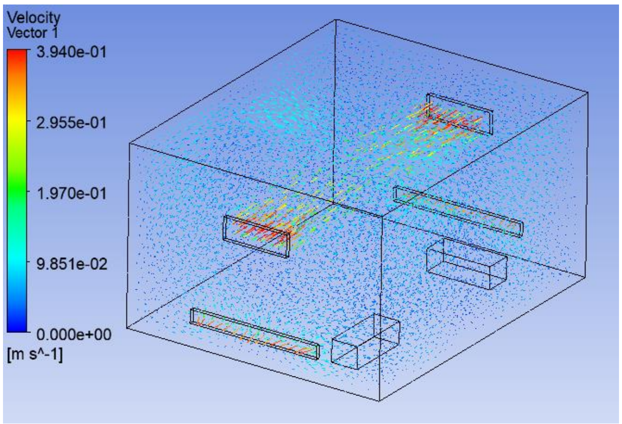
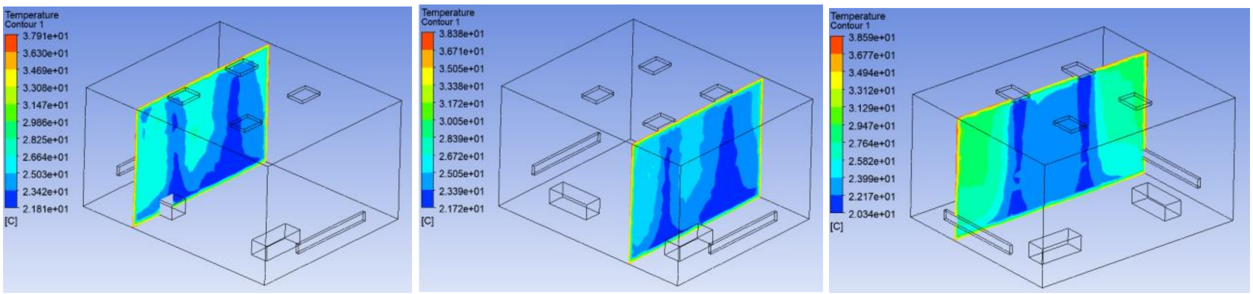


Figure 11 Velocity vector of case 3

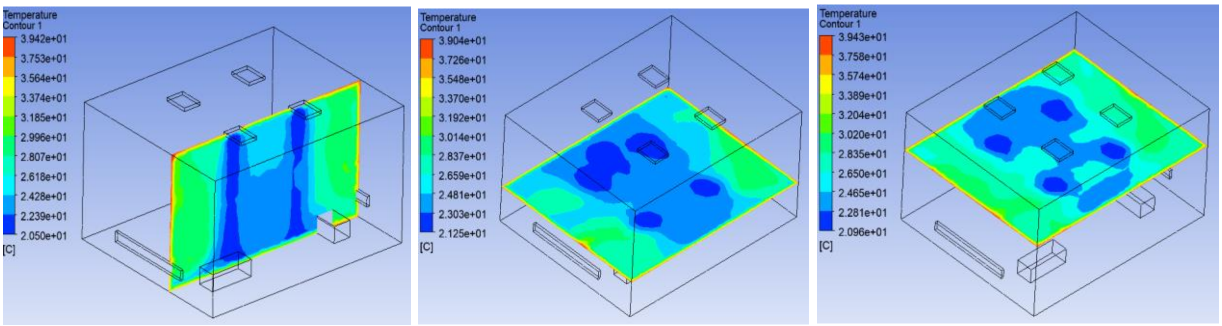
Case 4

Temperature contour

Air entered in the room at 20℃ and as far as reach increases in temperature due to the high room temperature. The figure shows that heat flux emitted from the floor surface disarranges the stratification of the temperature. It is evident from the figure that the highest air temperature in this case belongs to the air flow in vicinity of wall and floor surface (39 − 38°C) and that the airflow in vicinity of the outlet vent has the average air temperature (29 − 25°C). In the present section, effect of the floor temperature on the fresh air distribution in the breathing zone is evaluated. Temperature distribution are shown in figure in the patient room in different plans. In this case area of the inlet and outlet both area is increases. Change the location of the inlet and outlet do investigate the average room temperature. In that case, average room temperature is decrease from case 1. In this case average temperature at surrounded at patient is 26 − 22 ℃. In this case lowest temperature around patient as compared to the other case because inlet is placed at roof and outlet in both side walls.



(a) Plane 1 **(b) Plane 2** **(c) Plane 3**



(d) Plane 4 **(e) Plane 5** **(f) Plane 6**

Figure 12 Temperature contour of the case 3

Velocity vector

In this case at inlet air velocity is 0.39 m/s. it is in decreasing when spread in the room. When air strike at floor forms a vertex to all side. In this scenario fresh air is circulate all around patient and all the area of the room. At the outlet velocity is maximum which is 0.5 m/s but in the inlet, velocity is 0.39 m/s. the average velocity at room is between 0.25 – 0,125 m/s. In this case outlet are near to the patient bed so, air circulation is very well at around patient. Velocity vector show in figure below;

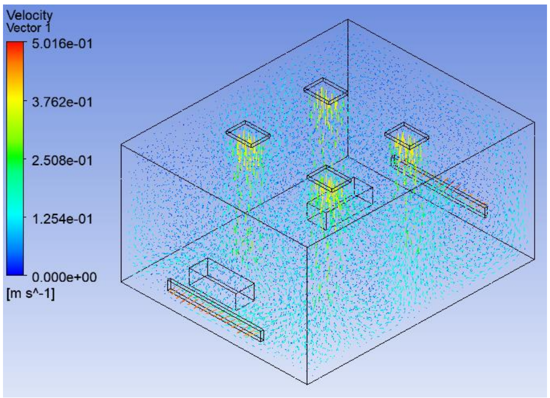


Figure 13 Velocity vector of case 4

Comparison of temperature in all cases

Compares the air temperature distribution over the all planes for various scenarios. High temperature near the wall due to the heat flux emitted from the wall and floor surface on every plane which is considered. Contour simply that the configuration of ventilation vents can affect significantly the temperature distribution and, particularly, the formation of plumes in the lower half of the room. It is evident from the figure that the highest air temperature in all cases belongs to the air flow in vicinity of the plane 3 for case 1 at 300 seconds (27.229 °C) and lowest air temperature in vicinity of the planes 6 for case 2 at 300 seconds (23.948 °C).

At 600 seconds it is evident from the figure the highest air temperature belongs to the air flow in vicinity of the plane 3 for case 1, (24.832 °C) and lowest air temperature in plane 6 case 2, (22.766 °C). Average room temperature at 300s and 600s is lowest in case 2 (25.258 °C) and case 4 (23.551 °C) respectively show in the figure.

Table 3 Average room temperature of all cases in °C

Time(s)	Case 1	Case 2	Case 3	case 4
300	26.037	25.258	25.772	25.56
600	24.014	23.572	23.842	23.551

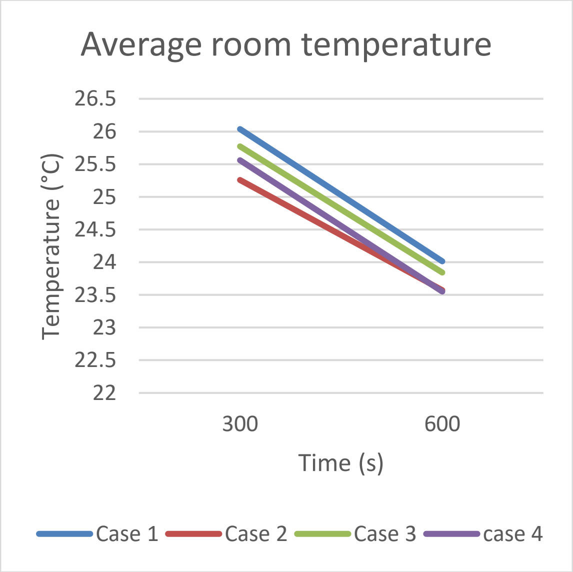


Figure 14 Average room temperature of all cases

Table 4 Temperature on all planes of all cases at 300s in °C

	Plane 1	plane 2	plane 3	plane 4	plane 5	plane 6
case 1	26.428	26.367	27.229	26.119	26.47	24.605
case 2	25.318	25.049	26.129	25.089	25.554	23.948
case 3	25.74	25.82	26.274	26.287	25.912	24.859
case 4	24.715	24.609	25.17	25.988	25.785	26.38

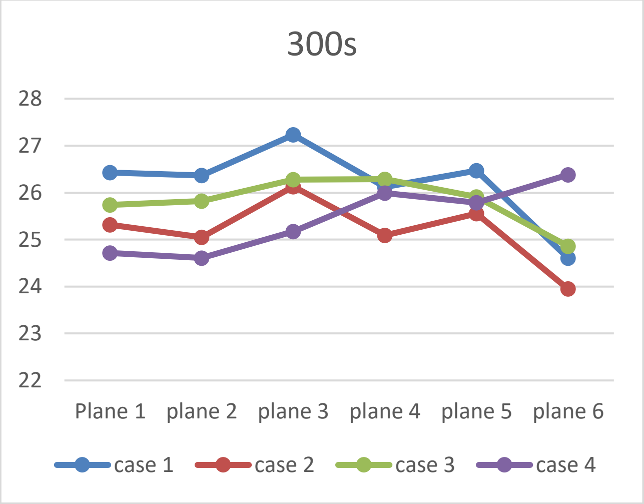


Figure 15 Temperature on all planes of all cases at 300s

Table 5 Temperature on all planes of all cases at 600s in °C

	Plane 1	plane 2	plane 3	plane 4	plane 5	plane 6
case 1	23.773	23.771	24.832	24.002	24.313	23.352
case 2	23.78	23.577	24.396	23.555	23.631	22.766
case 3	23.831	23.82	24.211	24.225	23.999	23.223
case 4	23.176	23.432	23.052	23.873	23.605	24.363

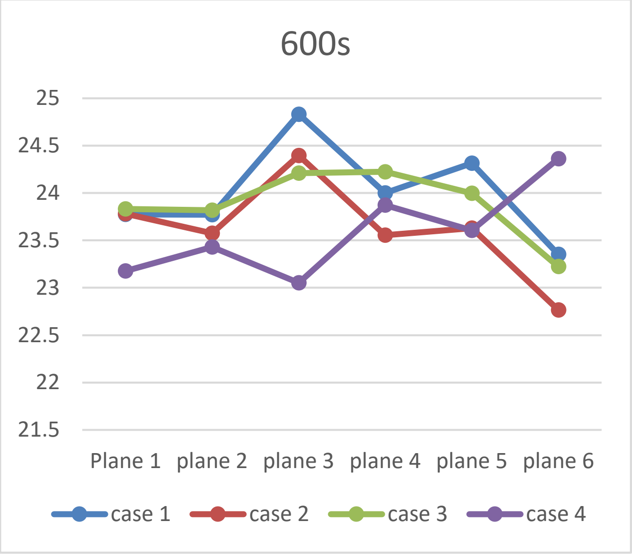


Figure 16 Temperature on all planes of all cases at 600s

Conclusion

In the present study, the distribution of fresh air in the class 3rd patient room of floor and wall heating systems under various ventilation scenarios was investigated. A computational fluid dynamic (CFD) model was established, to evaluate the turbulent airflow field, temperature distribution and Fresh air as well as to obtain relevant indicators for indoor air quality (IAQ). Four different ventilation scenarios were regarded in order to find the optimal ventilation design in terms of the better delivery of the fresh air. Furthermore, the integrated effects of the wall and floor temperature and inlet supply temperature along with the ventilation design on the distribution of Fresh air were addressed. The main findings of the present study based on the obtained results can be concluded as follows:

- The location of the air vents on the thermal comfort of occupants in the occupied zone it is recommended that the inlet diffuser should be located in the upper half wall.
- For a given wall and supply air temperature, increasing the ventilation rate enhanced the delivery of fresh air into the breathing zone for all ventilation scenarios.
- The height of the air vent outlet from ceiling to floor can affect the temperature gradient in the vertical direction, and thermal discomfort can be reduced by reducing the height of the vent location.
- The geometric design that is suitable for use in the class III inpatient room at Bitung Hospital is use in case 2, and 4, because geometry 4 has an air vent outlet height from the ceiling to the floor that is not too high, has an inlet diffuser placed on the upper half wall, and at roof of the room.

References

[1] R. T. Wang, J. C. Wang, and S. L. Chen, “Investigations on Temperatures of the Flat Insert Mold Cavity Using VCRHCS with CFD Simulation,” *Polymers (Basel).*, vol. 14, no. 15, 2022, doi: 10.3390/polym14153181.

[2] T. Lim and D. D. Kim, “Thermal Comfort Assessment of the Perimeter Zones by Using CFD Simulation,” *Sustainability*, vol. 14, no. 23, p. 15647, 2022, doi: 10.3390/su142315647.

[3] F. Hou, J. Ma, H. H. L. Kwok, and J. C. P. Cheng, “Prediction and optimization of thermal comfort, IAQ and energy consumption of typical air-conditioned rooms based on a hybrid prediction model,” *Build. Environ.*, vol. 225, no. November, 2022, doi: 10.1016/j.buildenv.2022.109576.

[4] F. Ascione, R. F. De Masi, M. Mastellone, and G. P. Vanoli, “The design of safe classrooms of educational buildings for facing contagions and transmission of diseases: A novel approach combining audits, calibrated energy models, building performance (BPS) and computational fluid dynamic (CFD) simulations,” *Energy Build.*, vol. 230, p. 110533, 2021, doi: 10.1016/j.enbuild.2020.110533.

[5] P. Patel, R. Karmur, G. Choubey, and S. Tripathi, *Recent Trends in Fluid Dynamics Research*, no. January. 2022. doi: 10.1007/978-981-16-6928-6.

[6] L. Li, Y. Wang, M. Wang, W. Hu, and Y. Sun, “Impacts of multiple factors on energy consumption of aging residential buildings based on a system dynamics model–Taking Northwest China as an example,” *J. Build. Eng.*, vol. 44, no. April, p. 102595, 2021, doi: 10.1016/j.jobe.2021.102595.

[7] M. Alkhalaf, A. Ilinca, and M. Y. Hayyani, “CFD Investigation of Ventilation Strategies to Remove Contaminants from a Hospital Room,” *Designs*, vol. 7, no. 1, 2023, doi: 10.3390/designs7010005.

[8] L. Lei, W. Chen, B. Wu, C. Chen, and W. Liu, “A building energy consumption prediction model based on rough set theory and deep learning algorithms,” *Energy Build.*, vol. 240, p. 110886, 2021, doi: 10.1016/j.enbuild.2021.110886.

[9] O. Bayer, “Paper_1.Pdf,” *SBIE - Simpósio Brasileiro de Informática na Educação*. pp. 257–272, 1947.

[10] S. Saran *et al.*, “Heating, ventilation and air conditioning (HVAC) in intensive care unit,” *Crit. Care*, vol. 24, no. 1, pp. 1–11, 2020, doi: 10.1186/s13054-020-02907-5.

[11] F. Zhang, N. Saeed, and P. Sadeghian, “Deep learning in fault detection and diagnosis of building HVAC systems: A systematic review with meta analysis,” *Energy AI*, vol. 12, no. January, p. 100235, 2023, doi: 10.1016/j.egyai.2023.100235.

[12] K. Sainath *et al.*, “Overview of ‘HVAC Systems’ in General Aspects,” *Int. J. Res. Appl. Sci. Eng. Technol.*, vol. 2, no. XII, pp. 558–569, 2014, [Online]. Available: www.ijraset.com

[13] F. N. Irani, M. Bakhtiaridoust, M. Yadegar, and N. Meskin, “A data-driven approach for fault diagnosis in multi-zone HVAC systems: Deep neural bilinear Koopman parity,” *J. Build. Eng.*, vol. 76, no. April, p. 107127, 2023, doi: 10.1016/j.jobe.2023.107127.

[14] W. Zheng *et al.*, “COVID-19 Impact on Operation and Energy Consumption of Heating, Ventilation and Air-Conditioning (HVAC) Systems,” *Adv. Appl. Energy*, vol. 3, no. May, p. 100040, 2021, doi: 10.1016/j.adapen.2021.100040.

[15] M. G. Albayati, J. De Oliveira, P. Patil, R. Gorthala, and A. E. Thompson, “A market study of early adopters of fault detection and diagnosis tools for rooftop HVAC systems,” *Energy Reports*, vol. 8, pp. 14915–14933, 2022, doi: 10.1016/j.egyr.2022.11.017.

[16] A. Jahanbin and G. Semprini, “On the optimisation of age of the air in the breathing zone of floor heating systems: The role of ventilation design,” *Energy Built Environ.*, no. July, 2022, doi: 10.1016/j.enbenv.2022.08.005.

[17] L. Liang *et al.*, “Comparative analysis of technical requirements for Heating, Ventilating, and Air Conditioning (HVAC) systems in high-biocontainment facility standards,” *Biosaf. Heal.*, vol. 5, no. 1, pp. 1–7, 2023, doi: 10.1016/j.bsheal.2022.12.003.

[18] T. Tsoutsos, S. Tournaki, C. A. De Santos, and R. Vercellotti, “Nearly zero energy buildings application in mediterranean hotels,” *Energy Procedia*, vol. 42, pp. 230–238, 2013, doi: 10.1016/j.egypro.2013.11.023.

[19] S. O. D. Niaki, M. Pourfallah, and A. Z. Ghadi, “Feasibility and investigation of residential HVAC system with combined ground source heat pump and solar thermal collectors in different climates of Iran,” *Int. J. Thermofluids*, vol. 20, no. July, p. 100427, 2023, doi: 10.1016/j.ijft.2023.100427.

[20] Y. D. Torres *et al.*, “Heating ventilation and air-conditioned configurations for hotelsan approach review for the design and exploitation,” *Energy Reports*, vol. 6, no. April, pp. 487–497, 2020, doi: 10.1016/j.egyr.2020.09.026.

[21] Q. Jiang, J. Chen, J. Hou, and Y. Liu, “Research on building energy management in HVAC control system for university library,” *Energy Procedia*, vol. 152, pp. 1164–1169, 2018, doi: 10.1016/j.egypro.2018.09.152.

[22] C. Becchio, S. P. Corgnati, M. Vio, G. Crespi, L. Prendin, and M. Magagnini, “HVAC solutions for energy retrofitted hotel in Mediterranean area,” *Energy Procedia*, vol. 133, pp. 145–157, 2017, doi: 10.1016/j.egypro.2017.09.380.

[23] Y. M. Abdullatif, A. Sodiq, T. Al-Ansari, N. N. Nassar, and A. I. Amhamed, “Optimizing chemisorption based direct air capture unit efficiency in HVAC systems: A study on the impact of DAC location and adsorption conditions as a response to the climate crisis and indoor air quality,” *Energy Convers. Manag.*, vol. 291, no. March, p. 117280, 2023, doi: 10.1016/j.enconman.2023.117280.

[24] U. B. E. S. C. F. D. Co-simulation, “Case Study : Impacts of Air-Conditioner Air Supply Strategy on,” 2023.

[25] A. Surapaneni and H. Company, “HVAC Design Considerations and Indoor Environmental Optimizations for a Cannabis Flowering Room,” no. April, 2023.

[26] H. Aulia, R. Budi, T. Faiz Dhiahaqi, and A. C. Ramadhani, “HVAC System Functionality with 3 Parameters such as Geometry Design,Duct Positions, and Cooling Time on Room Temperature Class III InpatientRoom at Bitung Hospital Based on Computational Fluid DynamicsSimulation,” no. January, 2023, [Online]. Available: https://www.researchgate.net/publication/367347494