Ventilation and Airflow Analysis in MFHT Room at IIT Bombay: Impact of Exhaust and AC, Comparison with Dual Exhaust Configuration

Deepanjan Das[1]

[1]M.Tech, Thermal and Fluids Engineering, IIT Bombay

Abstract

This study investigates the significance of Air Changes Per Hour (ACH) in understanding and managing indoor air quality. The focus lies on monitoring and optimizing ACH to ensure adequate ventilation, maintain indoor air quality, reduce the risk of disease transmission, and improve the well-being, comfort, and productivity of indoor occupants. The study analyses air residence time in a room equipped with one exhaust, one inlet (acting as a door), and one air conditioner. Through approximately 25 simulations with a fixed exhaust height and varying horizontal positions, the aim is to determine air residence time at two heights: 1.3 meters and 1.6 meters, corresponding to seated and standing individuals, respectively. The objective is to minimize air residence time by optimizing exhaust placement. Results indicate that residence time decreases further with two exhausts. Additionally, a machine learning algorithm achieves 92.16% and 86.10% accuracy in predicting air residence time at 1.6 meters and 1.3 meters, respectively, based on simulation data.

1. Introduction

The effective ventilation of indoor spaces plays a critical role in maintaining air quality and minimizing the risk of airborne disease transmission. The airflow patterns within a room are influenced by various factors, including the room's volume and the airflow rate through ducts and fans. Understanding these patterns is essential for assessing the effectiveness of air replacement and identifying areas where infectious particles may accumulate.

Key to this understanding is the layout of ventilation systems, particularly the positioning of ducts and vents relative to the room's layout. Airflow patterns created by these systems impact how aerosols spread within the space, with areas of concern often found in spots where air recirculates, such as room corners and around obstacles. These areas may allow infectious particles to accumulate, posing a risk to occupants.

Studying airflow patterns within rooms serves multiple purposes. Firstly, it aids in assessing the risk of airborne disease transmission by identifying potential accumulation zones. Additionally, it facilitates the design of efficient ventilation systems, enabling engineers to optimize layouts for the even distribution of fresh air and the removal of stale air. This optimization not only enhances indoor air quality but also minimizes energy consumption.

Furthermore, airflow patterns influence indoor air quality by affecting the distribution of pollutants and allergens. By understanding these patterns, steps can be taken to improve air quality in compromised areas. Moreover, airflow patterns impact occupant comfort by influencing factors such as temperature and humidity. Designing spaces with optimal airflow can create environments that promote productivity and well-being.

The simulations for a common case were conducted by the three of us: Akhilesh Ponkshe, Deepanjan Das, and Sayalee Kushire. Additionally, I conducted unique case simulations with two exhaust in the same room.

In summary, the study of airflow patterns in rooms is crucial for ensuring indoor environmental quality, optimizing ventilation systems, and mitigating the risk of airborne disease transmission. This research aims to investigate the ventilation dynamics of a room with exhaust positioned at the right corner, supplemented by an air conditioner on the right wall, and compare its effectiveness with a scenario involving two exhausts.

2. Problem Statement

Here, In this study the airflow pattern in a room setup, with a focus on recirculation zones. The geometry of the room is shown in Fig. 1

The room consists of a door (inlet), AC located on the right wall at the centre, and one exhaust located on the back wall. For studying air residence time, we have considered exhaust at a height of 3m from bottom. The height of the exhaust is kept fixed, and its position is varied on horizontal plane xz.

For different positions of exhaust on 3 different walls i.e. left wall, back wall, and right wall air residence time is calculated. Our flow is inlet driven. We have used the k-epsilon turbulence model for simulation. We have considered two different heights 1.3m and 1.6m, which is the average height of a person while sitting and standing. Aim to reduce air residence time at these heights by breaking the recirculation zone. It had been performed a total of 27 simulations and then divided our data into training and testing for the ML model. This will help us to get the location of exhaust for which get the lowest air residence time. According to results position of exhaust and AC will be decide.

Also, for comparison purpose a unique simulation was conducted in which another exhaust is added into the room. In unique case have one exhaust in front wall, and another exhaust at right wall at hight of 3m with regular intervention of split AC from right wall.

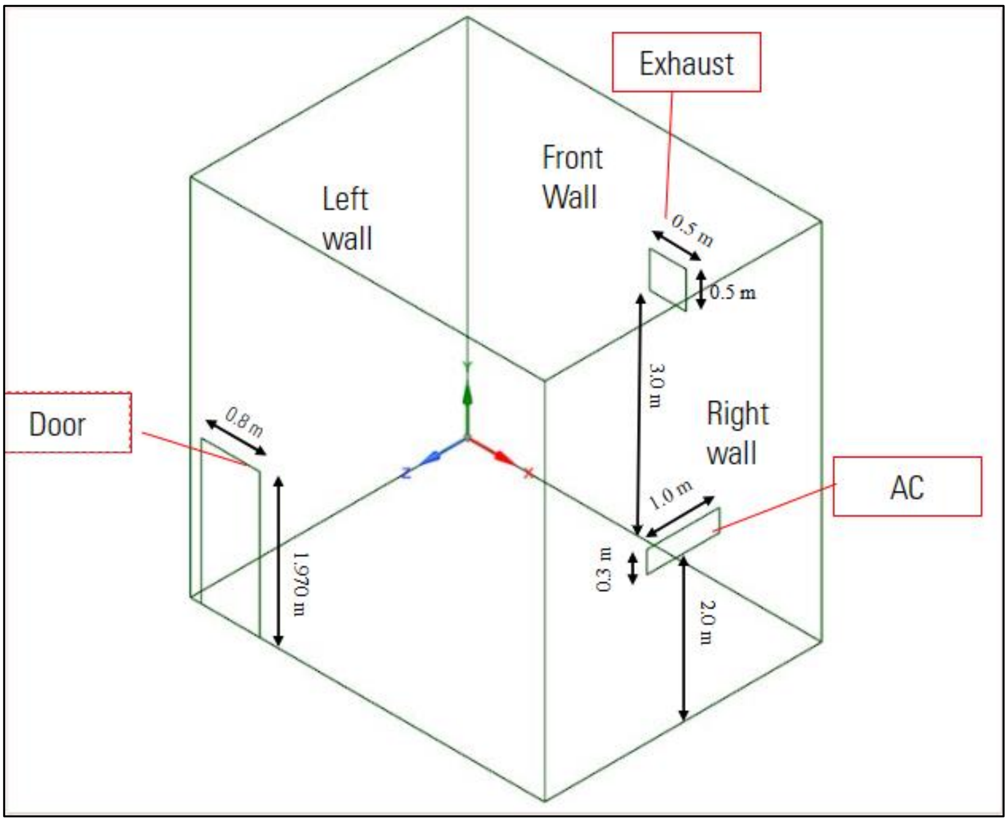


Figure 1 Geometry

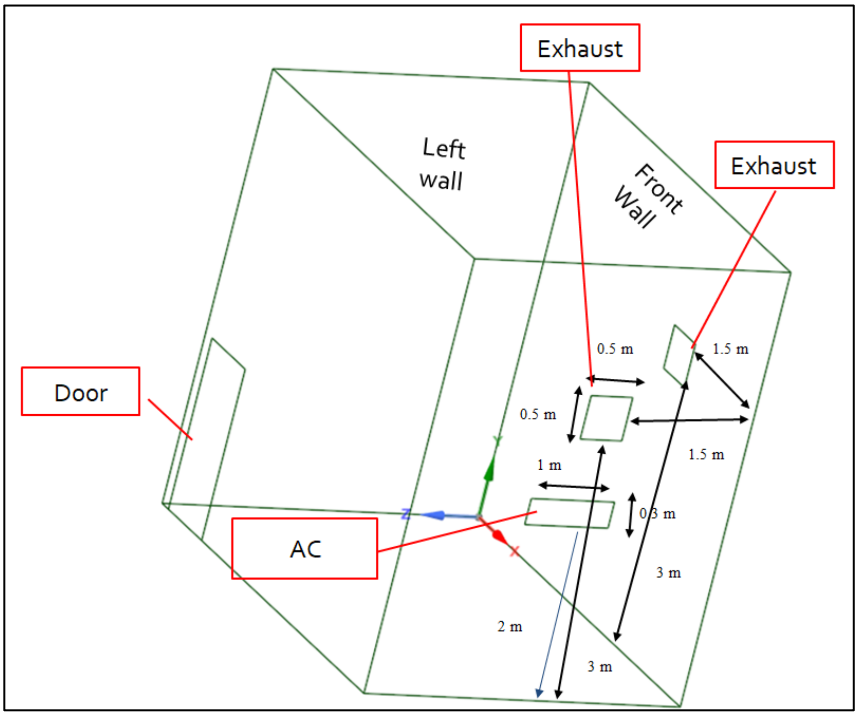


Figure 2 Geometry of unique case with two exhausts

3. Governing Equations

Fluid considered in this study is air, and air is assumed to be incompressible and ideal gas.

NS equations are

$$\nabla \cdot \boldsymbol{u} = 0 \qquad \textbf{(Eq. 1)}$$

$$\frac{D\boldsymbol{u}}{Dt} = -\frac{\nabla p}{\rho} + \nu \nabla^2 \boldsymbol{u} \quad \textbf{(Eq. 2)}$$

The General Scalar convection equation is given as

$$\frac{\partial \rho \emptyset}{\partial t} + \frac{\partial \rho u_j \emptyset}{\partial x_j} = S \;\; \textbf{(Eq. 3)}$$

Scalar convection for air residence time is

$$\frac{\partial \rho s}{\partial t} + \frac{\partial \rho u_j s}{\partial x_j} = 1 \;\; \textbf{(Eq.4)}$$

The necessary equations used for calculating turbulence are:

$\boldsymbol{k}$ = 3 /2 ($U_\infty \boldsymbol{I}$) ^2

$\boldsymbol{I} = \mathbf{0.16}(\boldsymbol{Re})$^ ($-\mathbf{1}/\ \mathbf{8}$)

$\boldsymbol{\varepsilon} = (\mathbf{0.164} \cdot (\boldsymbol{k})$^$\mathbf{1.5})/(\mathbf{0.07} \cdot \mathrm{L})$

Where:

$\boldsymbol{k}$ is turbulent kinetic energy,
$\boldsymbol{I}$ is the turbulent intensity

$\boldsymbol{\varepsilon}$ Turbulent dissipation rate

L is the inlet length

U_∞ is a characteristic velocity scale

4. Simulation Procedure

4.1 Geometry and Mesh

Geometry was created in Ansys. Meshing was also done in ANSYS. Details of the geometry are seen in Fig. 2. Fig. 3 shows the meshing. There are approximately 1,65,000 elements. Exhaust is mounted at fixed height of 3 m from the floor and AC is intervene from the right wall of the room. AC is mounted at fixed height of the 2 m from the floor.

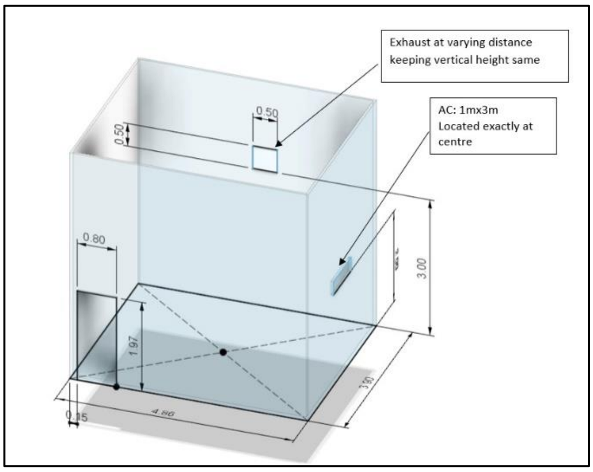


Figure 3 Geometry with dimensions

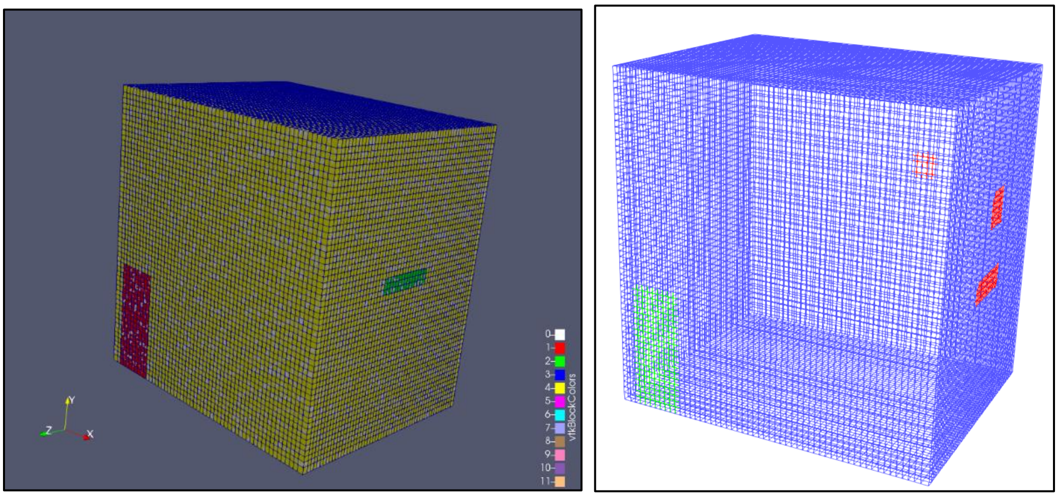


Figure 4 ANSYS Mesh

4.2 Initial and Boundary Conditions

Initial Conditions:

	U	P	k	epsilon
IC	internalField uniform (0 0 0)	internalField uniform 0	internalField uniform 9.7534e-5;	internalField uniform 2.82e-6

Boundary conditions:

	U	P	k	epsilon
inlet	Fixed value (-0.165m/s)	Zero gradient	turbulentIntensityKineticEnergyInlet 0.04887	turbulentMixingLengthDissipationRateInlet; 0.07
outlet	Zero Gradient	Fixed value 0	Zero Gradient	Zero Gradient
ac	Fixed Value	Zero Gradient	turbulentIntensityKineticEnergyInlet 0.04887	turbulentMixingLengthDissipationRateInlet; 0.07
walls	No slip	Zero Gradient	kqRWallFunction	epsilonWallFunction

Calculations:

AC inlet flow rate of 350 cubic feet per min was obtained, (florida solar energy center).

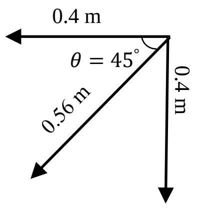
$$v_2 = \frac{350 \times 0.304^3}{60 \times 0.3 \times 1} = 0.56\, m/s$$

The exhaust outlet flow rate was taken as 26 $\frac{m^3}{min}$. (SHYUAN YA, Taiwan)

$$v_3 = \frac{26}{60 \times 0.5 \times 0.5} = 1.7\, m/s$$

AC having two components along x and y axis is considered. By providing velocity at AC and Exhaust, inlet velocity is calculated. Velocity has two components with $\theta = 45^{\circ}$, For AC velocity is given in an inclined manner in negative x and y direction.

It is provided as (-0.4 -0.4 0).



By continuity equation

$\rho_1 \times A_1 \times v_1 + \rho_2 \times A_2 \times v_2 = \rho_3 \times A_3 \times v_3$

$0.8 \times 1.970 \times v_1 + 1 \times 0.3 \times 0.56 = 0.5 \times 0.5 \times 1.7$

$v_1 = 0.165\, m/s$

$Re_1 = \frac{\rho \times U_\infty \times L}{\mu} = \frac{0.165 \times 0.8}{10^{-5}} = 13200$

$Re_2 = \frac{\rho \times U_\infty \times L}{\mu} = \frac{0.4 \times 0.3}{10^{-5}} = 12000$

$Re_1 > Re_2$ we have considered Re_1 for our calculations.

Turbulent Intensity, $I = 0.16 \times Re^{-0.125} = 0.04887$

Turbulent Kinetic Energy, $k = \frac{3}{2} \times (U_\infty \times I)^2 = 9.7534 \times 10^{-5}$

Turbulent Dissipation rate, $\epsilon = \frac{C_\mu^{\frac{3}{4}} \times k^{1.5}}{0.07 \times L} = 2.8 \times 10^{-6}$

Where,

$\boldsymbol{\rho}$ is the density of the fluid

$\boldsymbol{U_\infty}$ is the velocity of the fluid

$\boldsymbol{L}$ is the characteristic length of the domain.

$\boldsymbol{Re_1}$ is the Reynolds number based on the inlet of door domain

$\boldsymbol{Re_2}$ is the Reynolds number based on AC inlet

$\boldsymbol{C_\mu}$ is empirical constant its value is 0.09

4.3 Solver

Solver used for flow velocity simulation is simpleFoam solver which is steady state turbulence solver.

For solving general scalar convection equation, solver artFoam is developed. It solves source term “S”. It is subset of scalarTransportFoam with some special changes to solve scalar convection equation. In createFields “T” is substituted with source term “S”. Then for solving governing equation (eq. 3) rewrite fvScalarMatrix TEqn in scalarTransportFoam file as

fvm::ddt(S) + fvm::div(phi, S)- fvm::laplacian(DS, S) == sourceS

“artFoam” solver is a steady state to solve air residence time simulation.

5. Results and Discussions

❖ Result for common case

1) Air Flow Simulation results

In Figures 5 and 6, we present the computed flow field solutions displaying velocity vectors corresponding to varying exhaust positions along the walls. Figure 5(a) illustrates a horizontal plane positioned 1.3 meters above the floor, with the exhaust located on the front wall, 1.5 meters away from the left wall. Here, a predominant flow pattern is observed from the rear to the front, delineated between the door and the exhaust. Additionally, a dominant downward flow is evident from right to left between the air conditioner (AC) and the exhaust. Transitioning to Figure 5(c), the horizontal plane remains at 1.3 meters from the floor, while the exhaust position shifts to 2.8 meters from the left wall on the front wall. Analogous to the prior scenario, a prevailing flow direction is established between the door and the exhaust, proceeding from back to front. Similarly, a significant downward flow is noted between the AC and the exhaust, moving from right to left. Figure 5(b) explores an alternate situation with the exhaust placed on the left wall, 2.1 meters away from the back wall, while maintaining the horizontal plane at 1.3 meters from the floor. Here, a predominant flow develops from the rear to the front, observed between the door and the exhaust. Particularly noteworthy is the discernible recirculation zone adjacent to the AC, highlighted within a red box. Lastly, in Figure 5(d), the horizontal plane remains fixed at 1.3 meters from the floor, while the exhaust is positioned on the right wall, 1.2 meters from the front wall. As evident from preceding instances, a principal flow pattern emerges between the door and the exhaust, progressing from the rear to the front.

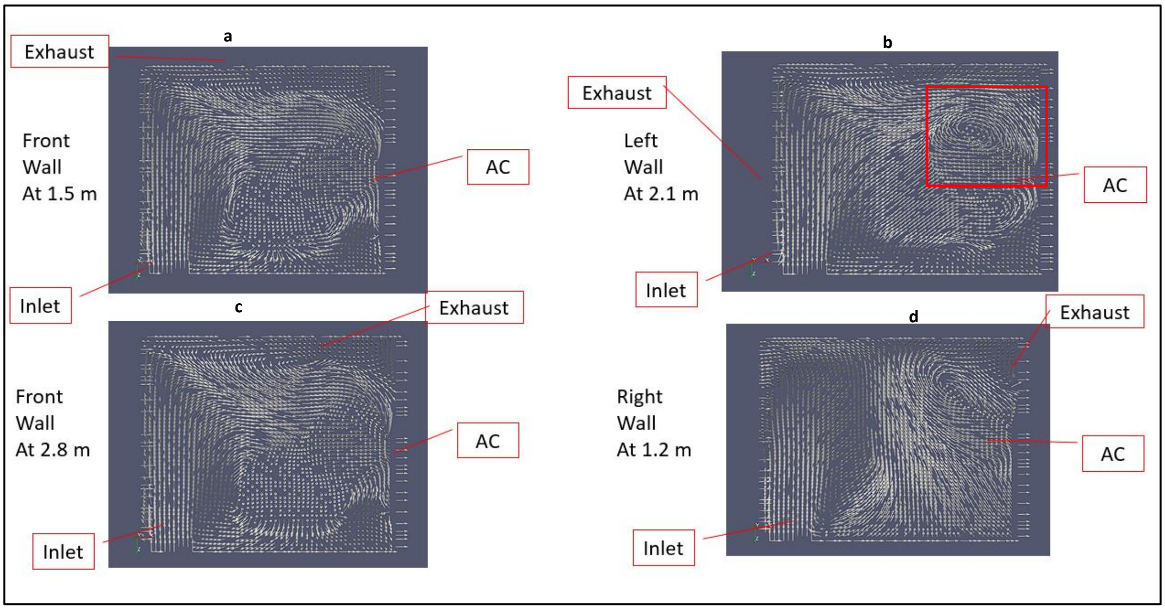


Figure 5 Velocity Vector at horizontal top views at height of 1.3 m from the floor for different position of exhaust in different wall of the room

In Figure 6(a), we explore a setup with a horizontal plane positioned 1.6 meters above the floor, featuring an exhaust on the front wall, 1.5 meters from the left wall. Primary airflow moves from back to front, creating flow between the door and exhaust. A downward primary airflow is observed between the AC and exhaust, flowing from right to left. Figure 6(c) mirrors this setup but with the exhaust 2.8 meters from the left wall. Again, primary airflow moves between the door and exhaust, while a downward flow occurs between the AC and exhaust. Transitioning to Figure 6(b), the horizontal plane remains at 1.6 meters, with the exhaust now on the left wall, 2.1 meters from the back. Primary flow moves from back to front, with a recirculation zone beside the AC. Lastly, in Figure 6(d), the horizontal plane stays at 1.6 meters, with the exhaust on the right wall, 1.2 meters from the front. Similar primary flow is observed between the door and exhaust, moving from back to front.

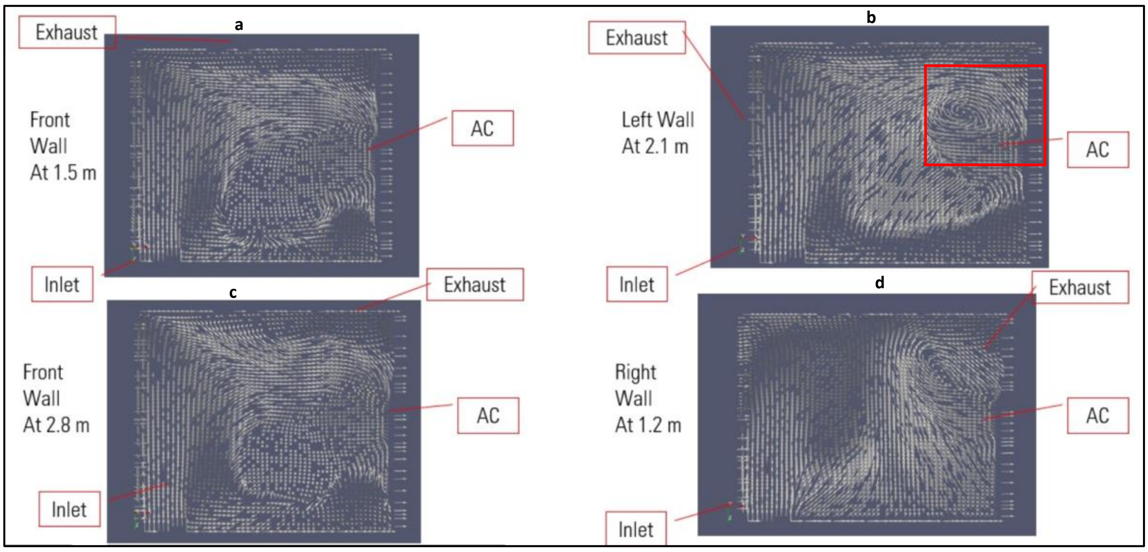


Figure 6 Velocity Vector at horizontal top views at height of 1.6 m from the floor for different position of exhaust in different wall of the room

2) Residence Time Simulation results

Case no.	x-coordinate	Y-coordinate	Z-coordinate	Face	y=1.3m	y=1.6m
1	0.2	3	3.8	Front	897.319	903
2	0.7	3	3.8	Front	922	927
3	1	3	3.8	Front	819	798
4	1.2	3	3.8	Front	870	875
5	1.5	3	3.8	Front	715	714
6	1.7	3	3.8	Front	561	561
7	2	3	3.8	Front	708	732
8	2.2	3	3.8	Front	757	833
9	2.5	3	3.8	Front	969	1044
10	2.7	3	3.8	Front	1021	1100
11	2.8	3	3.8	Front	889	939.78
12	3	3	3.8	Front	1079	1175
13	3.2	3	3.8	Front	1176	1208
14	3.5	3	3.8	Front	1277	1380
15	3.7	3	3.8	Front	1102	1114
16	4	3	3.8	Front	453	510
17	4.2	3	3.8	Front	447	467
18	4.86	3	3.6	Right	407	411
19	4.86	3	3.1	Right	408	379
20	4.86	3	2.8	Right	410	386
21	4.86	3	2.6	Right	391.5	370
22	4.86	3	2.3	Right	478	401
23	0	3	3.2	Left	829	826
24	0	3	3.1	Left	823	820
25	0	3	2.8	Left	1023	817
26	0	3	2.6	Left	1113	908
27	0	3	2.1	Left	757	752

Table 1: Number of simulations carried out maximum residence time with coordinate position of exhaust corner

The table illustrates various simulations conducted, displaying maximum air residence times at 1.3m and 1.6m heights for different exhaust positions. This data was pivotal for ML model training and testing. Notably, the orange-highlighted data reveals minimum air residence time when the exhaust is 1.7m from the left wall on the front wall. Conversely, the green-highlighted portion indicates significant dips in air residence time, potentially due to errors. Such data was omitted from ML model consideration. In fig. 7(a), we analyse a horizontal plane situated 1.3 meters above the floor, with the exhaust positioned on the front wall, 1.5 meters away from the left wall.it shows high residence time at the bottom right corner in the range of 400-800 seconds due to recirculation zones yet to be formed. In fig. 7(c) high residence time zone is shifted towards the door it is compacted to smaller area but more pronounced effect is observed. In fig 7(b), similar to 7(a), residence time zone is observed near the right bottom corner, but profound zone is seen. In fig 7(d), residence time zone is very less as compared to other position of exhaust. Highest residence time in that contour is about 200-400 seconds.

For fig 8. Closely similar results are observed 1.6 m plane with respect to exhaust position in 1.3 m plane from the floor.

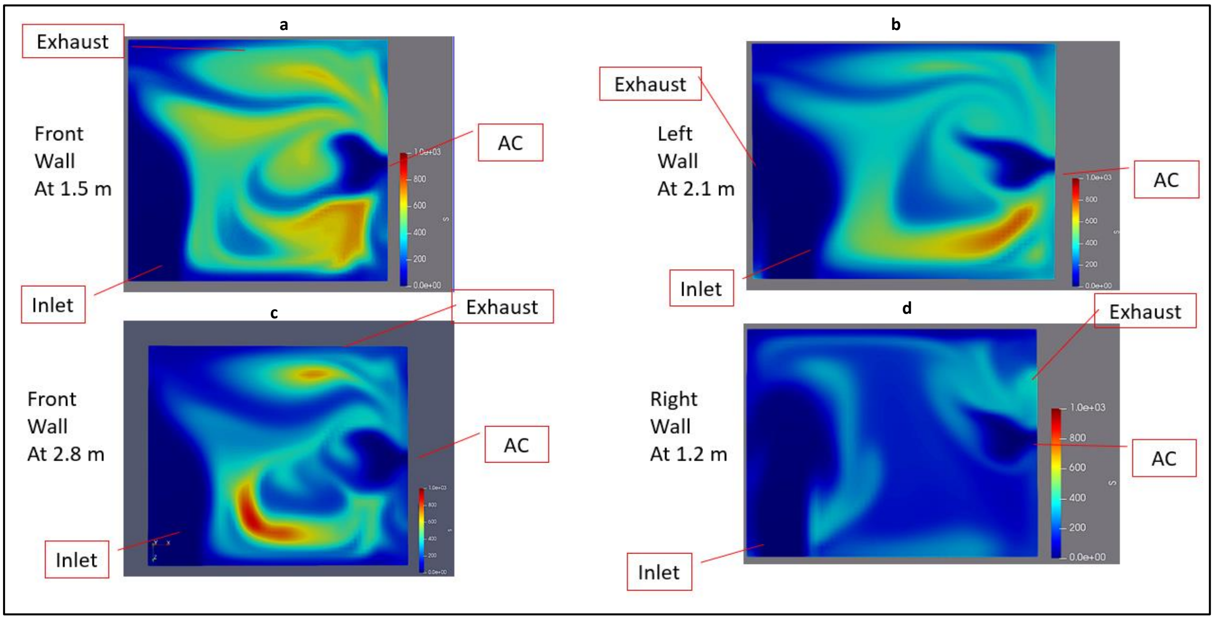


Figure 7 Air residence contours at horizontal top views at height of 1.3 m from the floor for different position of exhaust in different wall of the room

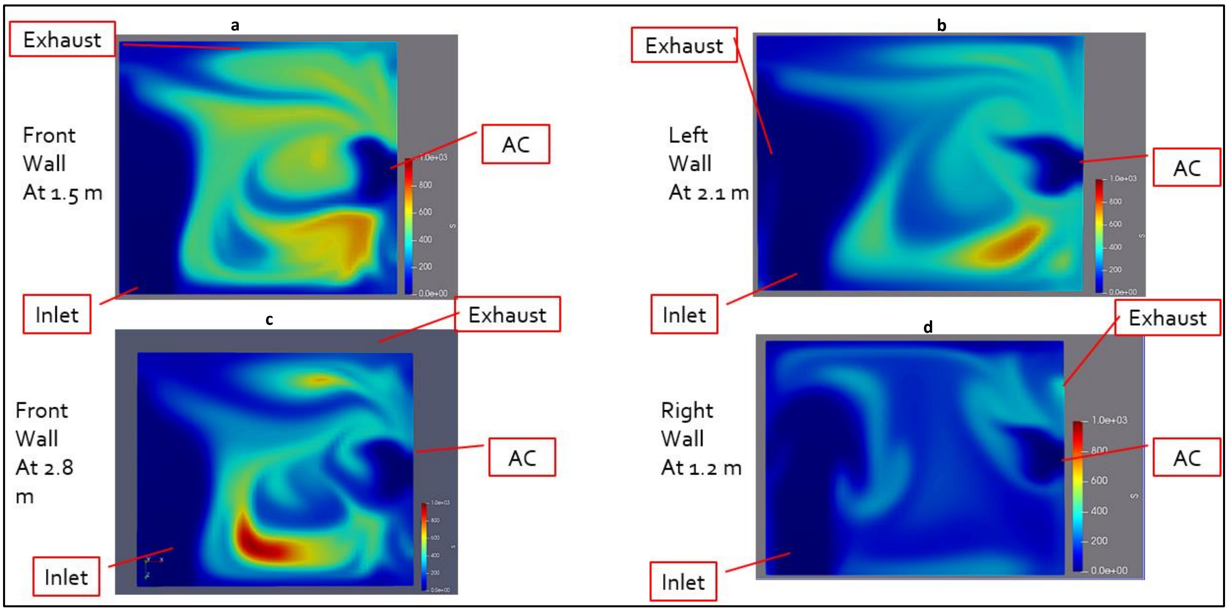


Figure 8 Air residence contours at horizontal top views at height of 1.6 m from the floor for different position of exhaust in different wall of the room

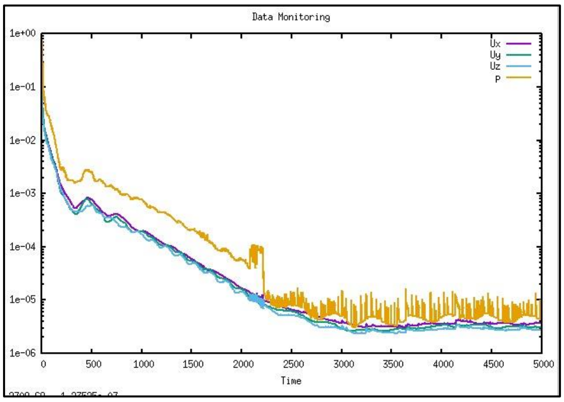


Figure 9 Residue plot for velocity and Pressure simulation

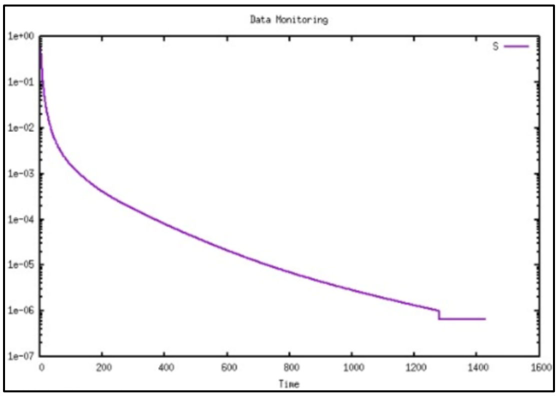


Figure 10 Residue plot for residence time simulations

3) Prediction of air residence time with the help of ML algorithm

We have used 20 training data and 5 testing data. There are 500 Epochs, 5 hidden layers and 256 neurons for training algorithm. MLP (Multilayer Perceptron) model was used in training. Geometric co-ordinates (X and Z) and maximum air residence value at 1.3 and 1.6 m were input values for training the data set. NRMSE (Normalised root mean square error), mean absolute error (MAE), R2 error were used as a loss function for prediction of ML algorithm.

For these small number of data set accuracy of prediction model based judged by NRMSE error, particularly in the context of regression problems. So, NRMSE was used in the code.

For Fig. 10 Accuracy ~ 86%, Normalised root mean square error ~ 15 %; For Fig. 11 Accuracy ~ 93%, Normalised root mean square error ~ 10 %

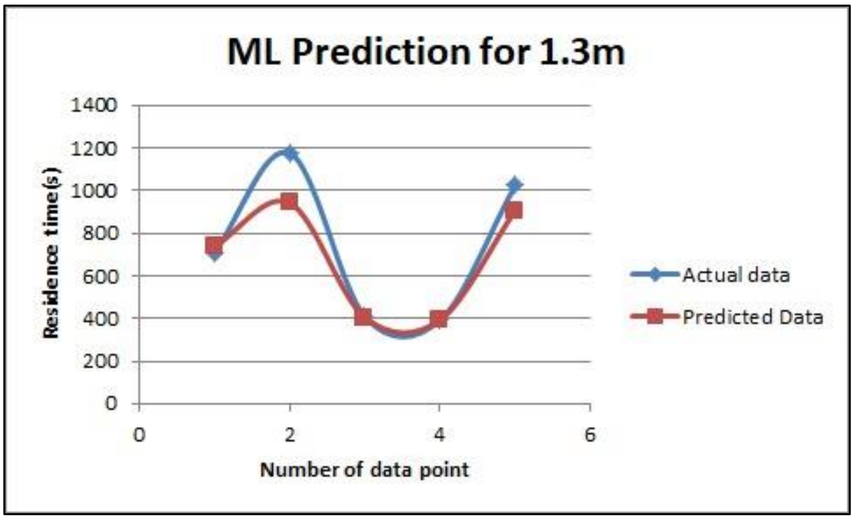


Figure 11 ML Prediction comparison with simulations for 1.3 m plane

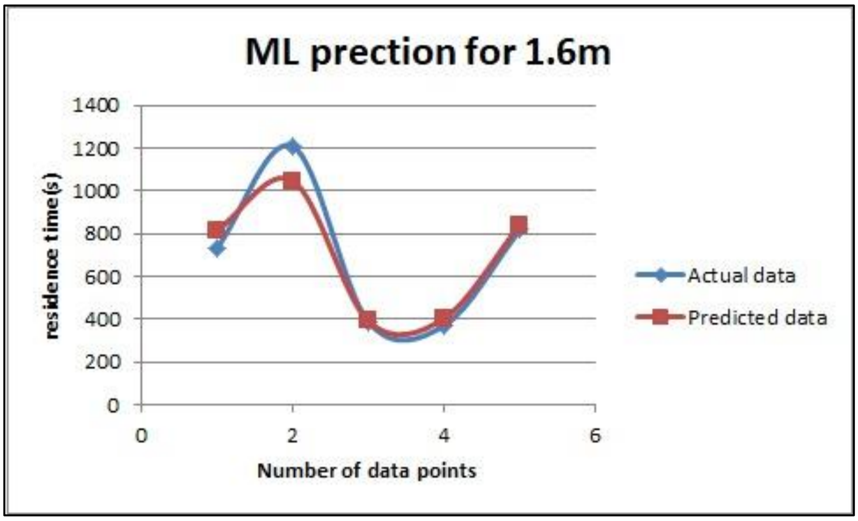


Figure 12 ML Prediction comparison with simulations for 1.6 m plane

❖ Result for unique case with two exhaust

1) Air Flow Simulation results

The motive of the unique case is to break the recirculation zone at the top corner. That's why I chose the exhaust position in the top corner of the room. In Figure 13a and 13c, vector plots at different heights from the floor of the MFHT room can be visualized. There are no minor changes observed in the heights. A small recirculation zone is formed at the top corner of the room beside the AC. Air residence time plots are visible at different heights: 1.3m and 1.6m from the ground in figures 13b and 13d, respectively. Residence time in the contour plot is distributed in a small range of 220 to 500 seconds.

In our previous results, we achieved a lower residence time of about 700 seconds approximately, which clearly shows that this unique case has better results. With the inclusion of the exhaust in the front wall along with the right wall, the maximum air residence time drastically reduces due to the disruption of the flow path of the AC. The AC has a downward inclined input velocity, which may also have some upward component due to the exhaust position above the AC. Hence, the flow will be distributed over the room greatly compared to our team's case. This case has a maximum air residence time of 465 seconds at 1.3m and 395 seconds at 1.6m. Figure 15 shows the volume integral of air residence time, indicating that this simulation reached a steady state.

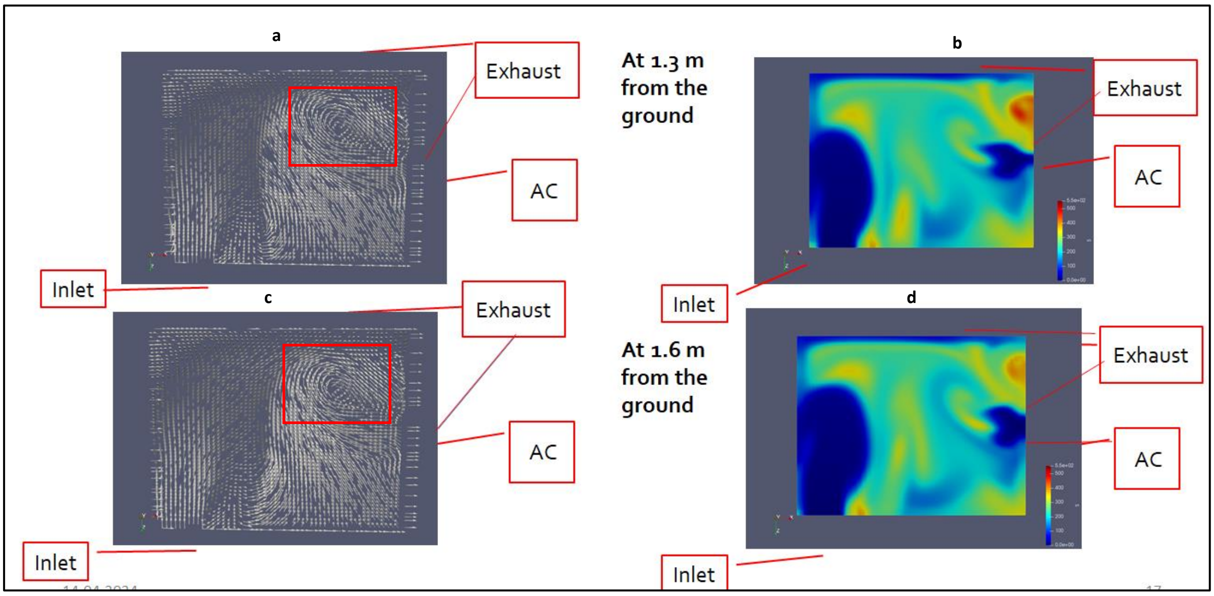


Figure 13: velocity vector plot and air residence time contour for unique case in at two different horizontal heights

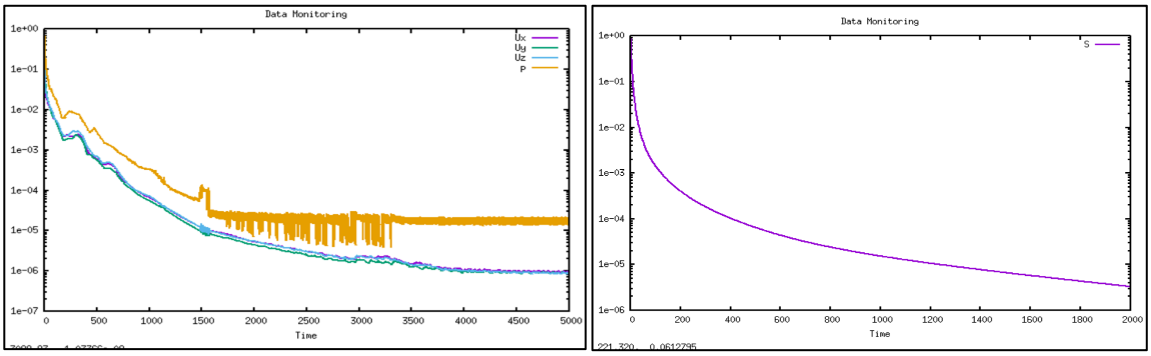


Figure 14 residue for pressure and velocities for unique case velocity simulation and air residence time(S)

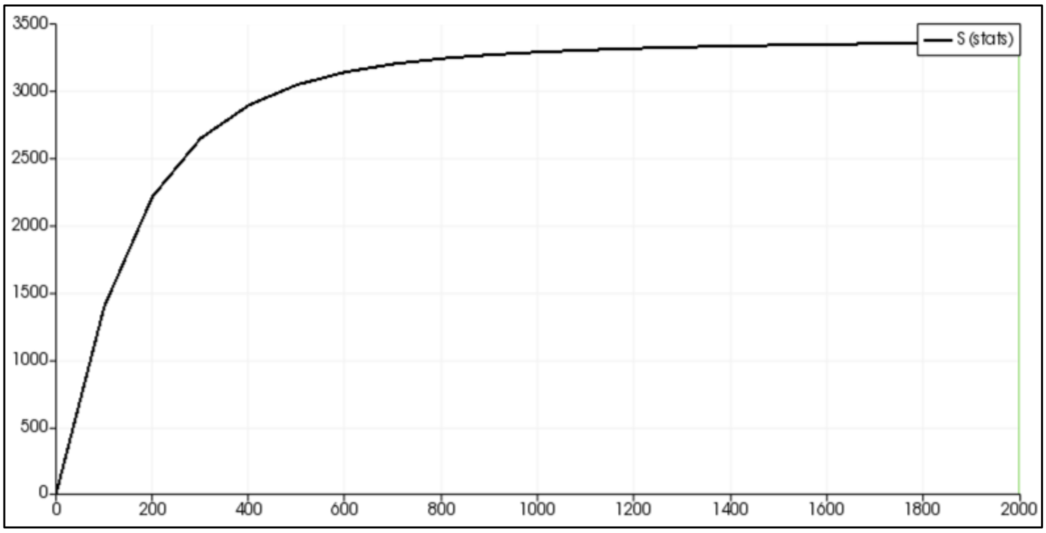


Figure 15 Volume integral of air residence time

Conclusion

- In Fig 16, as the exhaust location on the front wall shifts towards the right side, air residence time initially decreases, then increases again. This change might be due to the inlet flow being opposed by the AC flow.
- In Fig 17, closer to the AC on the right wall, air remains for a longer time compared to the area near the exhaust, which is farther away from the AC unit. This occurs because the airflow is drawn towards the exhaust system.
- In Fig 18, on the left wall, air residence time is significantly higher. This may be because the air entering from the door is drawn away by the exhaust.
- From the unique case simulation, it could be beneficial to use two lower-power exhausts instead of one high-power exhaust to maintain a good amount of air circulation in the room. This creates good ventilation, reducing the probability of getting infected by air bone diseases or viruses.
- ML algorithm can be used to predict the maximum air residence time accurately. We recommend using window AC or two exhausts in the room to reduce maximum air residence time and increase air ventilation rate.
- For our cases by calculation we get Air changing per hour (ACH) 18.19 $\frac{1}{hr}$, which tells the average residence time in our Case is about 200 sec.

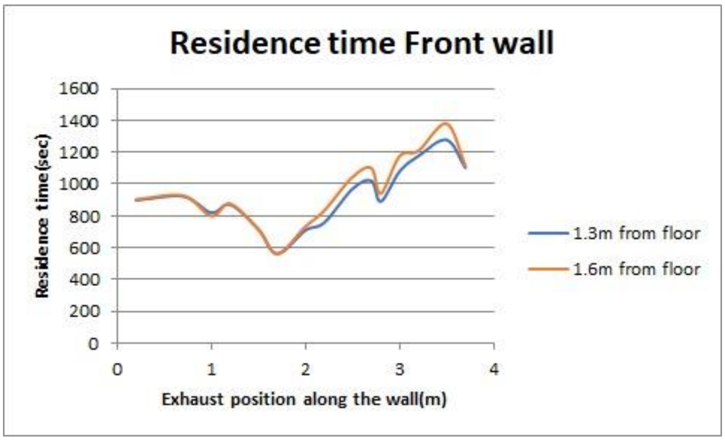


Figure 16 Residence time variations with exhaust position in front wall

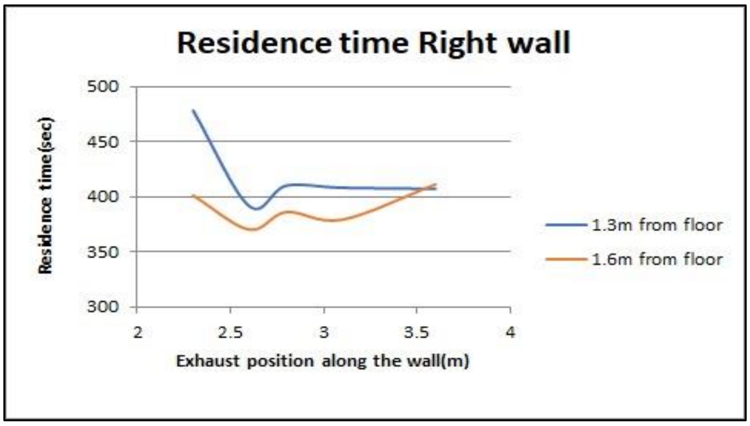


Figure 17 Residence time variations with exhaust position in right wall

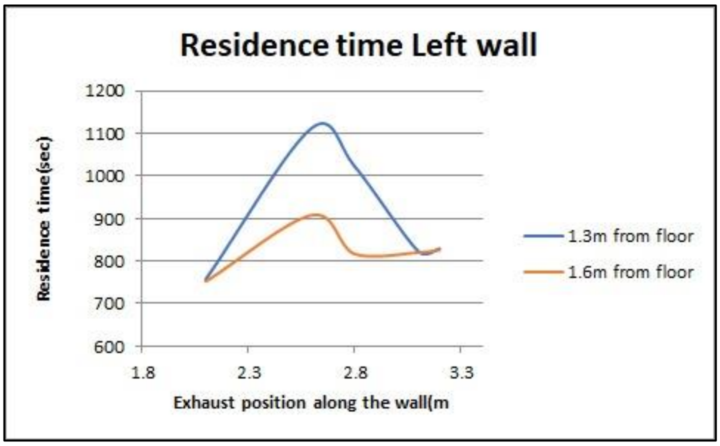


Figure 18 Residence time variations with exhaust position in left wall

References

Sinha, K., Yadav, M. S., Verma, U., Murallidharan, J. S., & Kumar, V. (2021). Effect of recirculation zones on the ventilation of a public washroom. *Physics of Fluids*, *33*(11).