Study of airflow in MFHT Room and impact of Exhaust and AC on air ventilation

Akhilesh Ponkshe[1]

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

Abstract

This research delves into the importance of Air Changes Per Hour (ACH) in comprehending and managing indoor air quality. The main focus is on monitoring and optimizing ACH to ensure sufficient ventilation, maintain indoor air quality, decrease the likelihood of disease transmission, and enhance the well-being, comfort, and productivity of indoor occupants. The study examines air residence time in a room equipped with one exhaust, one inlet (which functions as a door), and one air conditioner. Through approximately 27 simulations with a fixed exhaust height and different horizontal positions, the aim is to determine air residence time at two heights: 1.3 meters and 1.6 meters, corresponding to individuals seated and standing, respectively. The objective is to reduce air residence time by optimizing the placement of the exhaust. Results demonstrate that residence time decreases further by adding an extra window AC. Moreover, a machine learning algorithm attains 92.16% and 86.10% accuracy in forecasting air residence time at 1.6 meters and 1.3 meters, respectively, based on simulation data.

1. Introduction

The effective ventilation of indoor spaces is pivotal in upholding air quality standards and mitigating the risk of airborne disease transmission. The circulation of air within a room is subject to various factors, including its volume and the airflow rate facilitated by ducts and fans. Grasping these airflow patterns is crucial for evaluating the efficacy of air circulation and pinpointing zones where airborne particles may linger.

Central to this comprehension is the configuration of ventilation systems, especially the placement of ducts and vents in the room's layout. The airflow dynamics orchestrated by these systems impact the dispersion of aerosols within the space, with areas of concern often emerging where air stagnates, like room corners and around obstructions. Such zones may serve as hotspots for the accumulation of infectious particles, thereby posing a potential threat to occupants.

Analyzing airflow patterns within rooms serves a dual purpose. Firstly, it aids in assessing the vulnerability to airborne disease transmission by pinpointing potential zones of particle buildup. Additionally, it facilitates the design of efficient ventilation setups, empowering engineers to optimize layouts for uniform air distribution and the elimination of stagnant air. This optimization not only enhances indoor air quality but also minimizes energy usage.

Moreover, airflow patterns exert a direct influence on indoor air quality by shaping the dispersal of pollutants and allergens. A comprehensive understanding of these patterns enables targeted measures to enhance air quality in compromised areas. Furthermore, airflow dynamics impact occupant comfort by influencing factors like temperature and humidity. Designing spaces with well-regulated airflow fosters environments conducive to productivity and overall well-being.

We three(Akhilesh Ponkshe, Deepanjan Das, and Sayalee Kushire)[1] have performed this simulations for a common case. I have performed unique case simulations.

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 an exhaust positioned at the front wall, supplemented by a window air conditioner on the front wall, and one split AC on the right wall and compare its effectiveness.

2. Problem Statement

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

[1] IITB Student

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 the bottom.

The height of the exhaust is kept fixed, and its position is varied on the 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 the results position of the exhaust and AC will be decided.

Also, for comparison purposes, a unique simulation was conducted. In the unique case have one exhaust in the front wall, regular intervention of split AC from the right wall with extra Window AC in the front wall. (Figure 2)

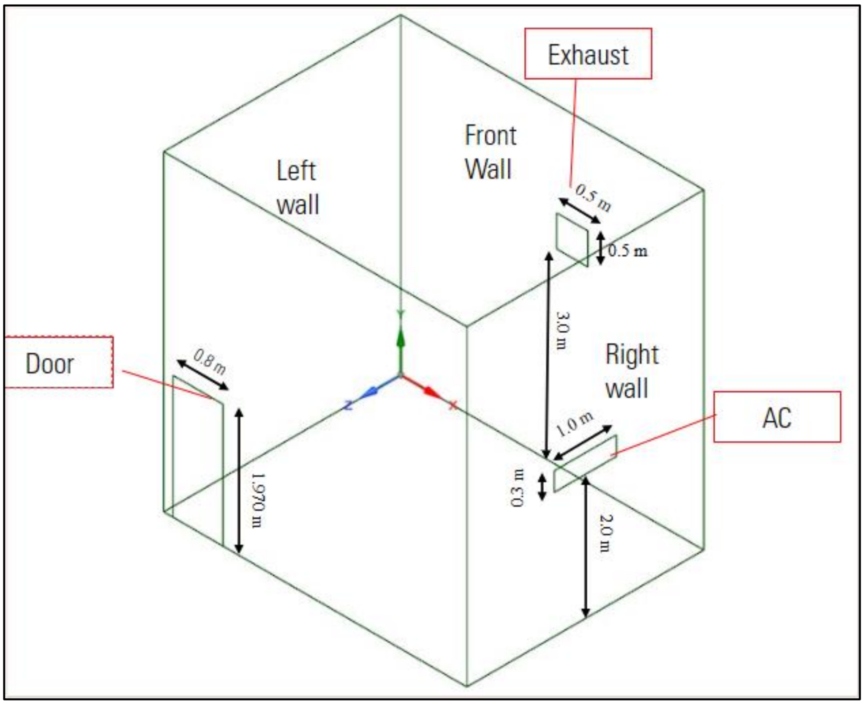


Figure 1 Geometry of common case

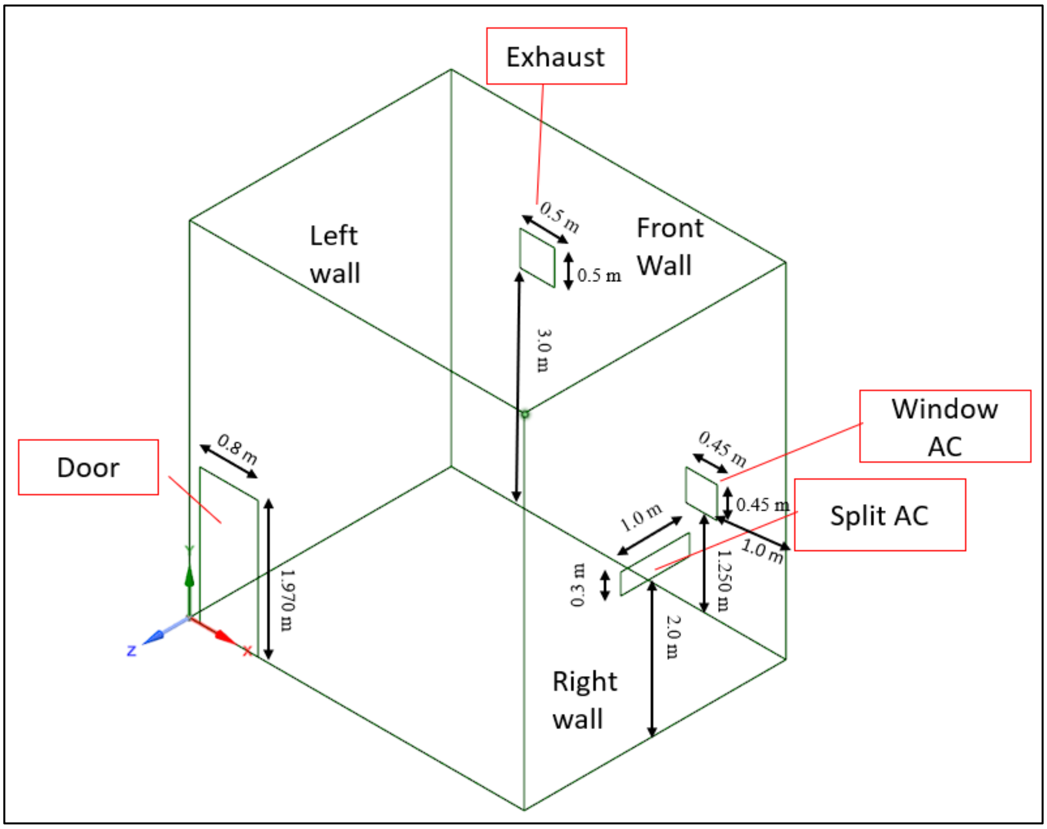


Figure 2 Geometry of unique case

3. Governing Equations

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

NS equations are

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

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

The General Scalar convection equation is given as

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

Scalar convection for air residence is given as

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

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 a fixed height of 3 m from the floor and AC is intervened from the right wall of the room. AC is mounted at a fixed height of 2 m from the floor.

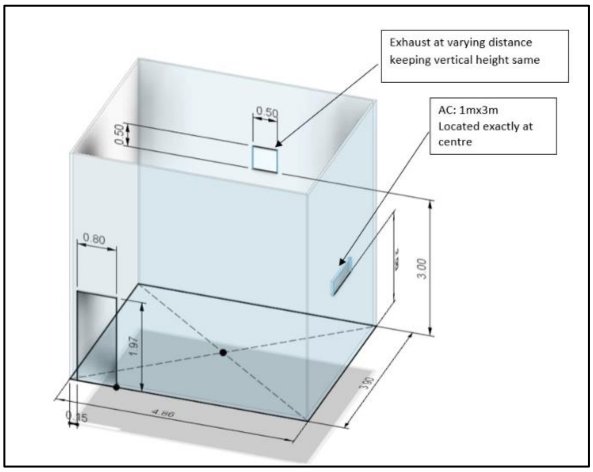


Figure 3 The General Scalar convection equation is given as

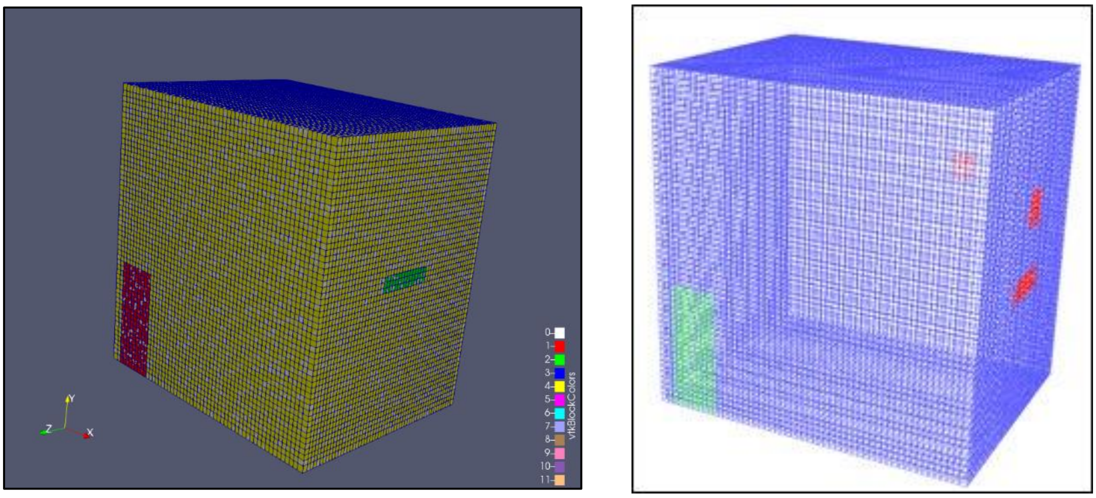


Figure 4 ANSYS Mesh

4.2 Initial and Boundary Conditions

Initial Conditions:

	U	P	k	epsilon
IC	internalField uniform (0 0 0)	internal field 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 for the general case:

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

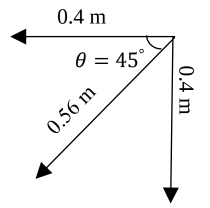
$$v_2 = \frac{350 \times 0.304^3}{60 \times 0.3 \times 1} = 0.56\ m/s$$

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

Calculations for a unique case

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

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

I obtained window AC flow rate as 440 $\frac{m^3}{hr}$. (Lloyd EW Series GLW12C2YWSEW)

$v_3 = \frac{440}{3600 \times 0.45 \times 0.45} = 0.6\ m/s$

$Re_3 = \frac{\rho \times U_\infty \times L}{\mu} = \frac{0.6 \times 0.45}{10^{-5}} = 27000$

$Re_3 > Re_1, Re_2$ have considered Re_3 for calculations.

Turbulent Intensity, $I = 0.16 \times Re^{-0.125} = 0.044689$

Turbulent Kinetic Energy, $k = \frac{3}{2} \times (U_\infty \times I)^2 = 1.078 \times 10^{-3}$

Turbulent Dissipation rate, $\epsilon = \frac{C_\mu^{\frac{3}{4}} \times k^{1.5}}{0.07 \times L} = 1.846 \times 10^{-4}$

$\boldsymbol{Re_3}$ is the Reynolds number based on window AC inlet, Here I have considered inlet velocity same as the common case, exhaust velocity will be changed accordingly.

4.3 Solver

The solver used for flow velocity simulation is simpleFoam solver which is a steady-state turbulence solver.

For solving general scalar convection equations, solver artFoam is developed. It solves the source term “S”. It is a subset of scalarTransportFoam with some special changes to solve the 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
```

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

5. Results and Discussions

- For common case

1) Air Flow Simulation results

In Figures 5 and 6, we present the computed flow field solutions showcasing velocity vectors at various exhaust positions along the walls. In Figure 5(a), we analyze 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. Here, a primary flow is established from back to front between the door and the exhaust. Additionally, there's a primary downward flow observed from right to left between the air conditioner (AC) and the exhaust. Moving to Figure 5(c), we maintain the horizontal plane at 1.3 meters from the floor, while shifting the exhaust position to 2.8 meters from the left wall on the front wall. Similar to the previous case, a primary flow is formed between the door and the exhaust, moving from back to front. Similarly, a primary downward flow is observed between the AC and the exhaust, flowing from right to left. Figure 5(b) explores another scenario where the exhaust is located 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 primary flow is established from back to front between the door and the exhaust. Notably, a recirculation zone is visible beside the AC, highlighted by a red box. Lastly, in Figure 5(d), the horizontal plane remains at 1.3 meters from the floor, while the exhaust is positioned on the right wall, 1.2 meters from the front wall. As seen in previous cases, a primary flow is established between the door and the exhaust, moving from back to front.

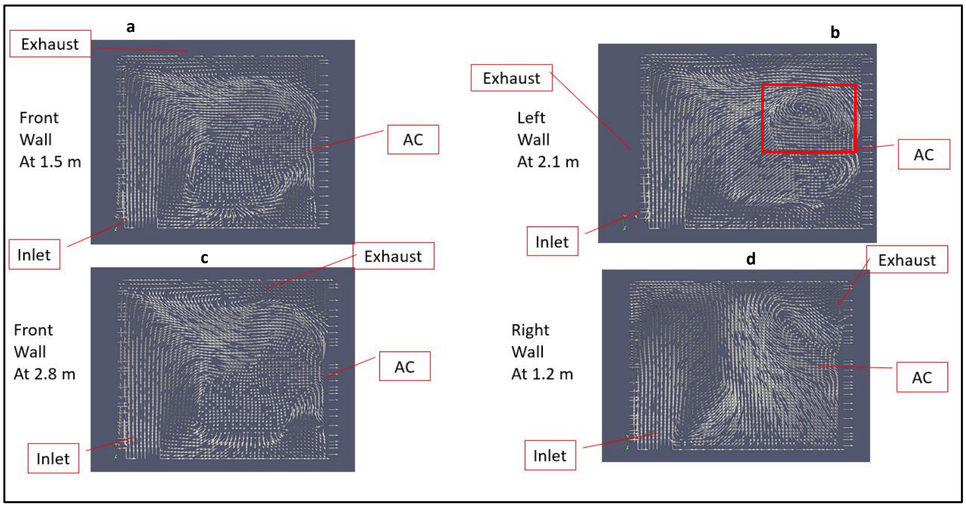


Figure 5 Velocity Vector at horizontal top views at a height of 1.3 m from the floor for different positions of exhaust in the different walls of the room

In Figure 6(a), we analyse a scenario where a horizontal plane situated 1.6 meters above the floor is under consideration. Here, the exhaust is positioned on the front wall, 1.5 meters away from the left wall. A primary airflow is established, moving from the back towards the front, creating a flow between the door and the exhaust. Additionally, there's a downward primary airflow, flowing from right to left, between the air conditioner (AC) and the exhaust. Figure 6(c) presents a similar setup, with the horizontal plane at 1.6 meters from the floor and the exhaust placed on the front wall, but this time 2.8 meters from the left wall. Again, a primary airflow is formed between the door and the exhaust, moving from back to front. Similarly, a downward primary airflow is observed between the AC and the exhaust, flowing from right to left. Moving to Figure 6(b), we examine another scenario where the horizontal plane is set at 1.6 meters above the floor. The exhaust is now positioned on the left wall, 2.1 meters away from the back wall. Here, a primary flow is established between the door and the exhaust, flowing from back to front. Notably, a recirculation zone appears beside the AC, indicated by a red box. Lastly, in Figure 6(d), the horizontal plane remains at 1.6 meters from the floor, but the exhaust is now located on the right wall, 1.2 meters from the front wall. Similar to the previous cases, a primary flow is formed between the door and the exhaust, moving from back to front.

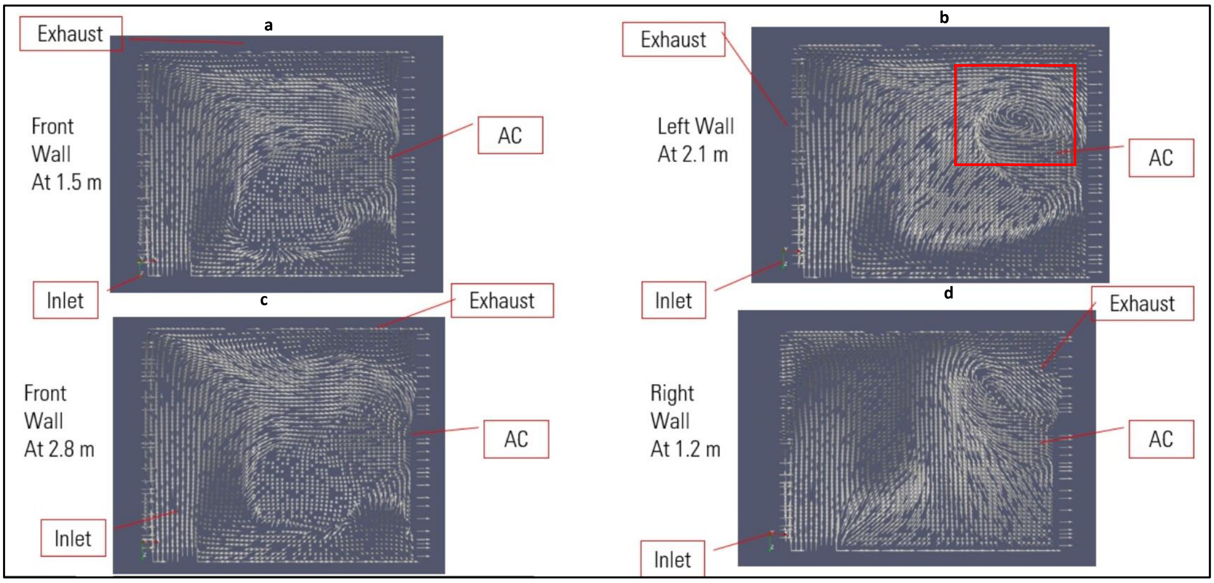


Figure 6 Velocity Vector at horizontal top views at a height of 1.6 m from the floor for different positions of exhaust in the different walls of the room

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, yellow colour showing minimum air residence time

Above is the table depicting the number of different simulations performed. It shows the values of maximum air residence time obtained at heights of 1.3m and 1.6m for varying positions of exhaust, the heights considered for analyzing the air residence time. This data was considered for ML training and testing. The yellow highlighted portion shows that we got minimum air residence time when the exhaust is located on the front wall at 1.7m from the left wall. This data was excluded while considering ML training and testing.

In Fig. 7(a), we examine a horizontal plane situated 1.3 meters above the floor, where the exhaust is positioned on the front wall, and 1.5 meters away from the left wall. It depicts elevated residence times at the bottom right corner, ranging from 400 to 800 seconds, attributed to yet-to-be-formed recirculation zones. In Fig. 7(c), the high residence time zone shifts toward the door, occupying a more compact area but exhibiting a more pronounced effect. Fig. 7(b) displays a similar pattern to Fig. 7(a), with a residence time zone near the bottom right corner, but with a more prominent effect observed. In Fig. 7(d), the residence time zone is significantly reduced compared to other exhaust positions, with the highest residence time in that contour ranging from 200 to 400 seconds.

For Fig. 8, closely similar results are observed at the 1.6-meter plane relative to the exhaust position in the 1.3-meter plane from the floor.

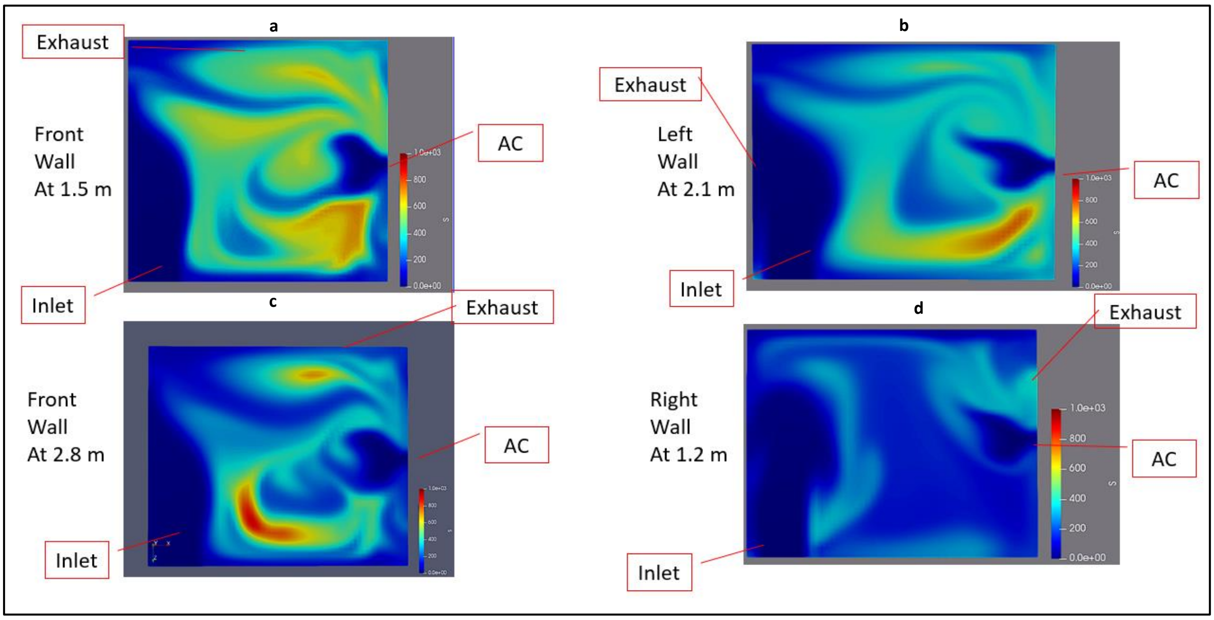


Figure 7 Air residence contours at horizontal top views at a height of 1.3 m from the floor for different positions of exhaust in different walls of the room

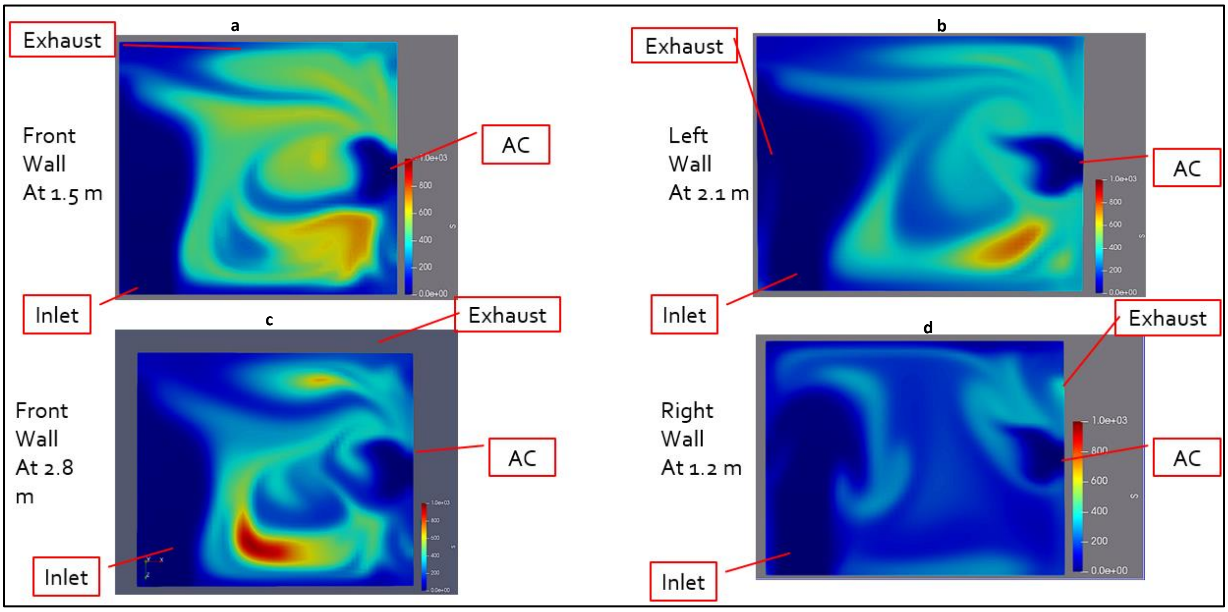


Figure 8 Air residence contours at horizontal top views at a height of 1.6 m from the floor for different positions of exhaust in the different walls of the room

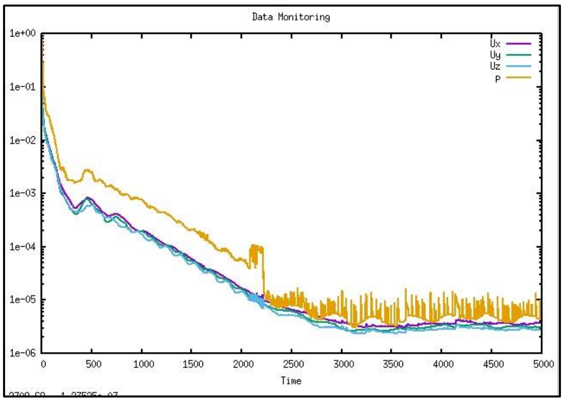


Figure 9 Residue plot for velocity and Pressure simulation

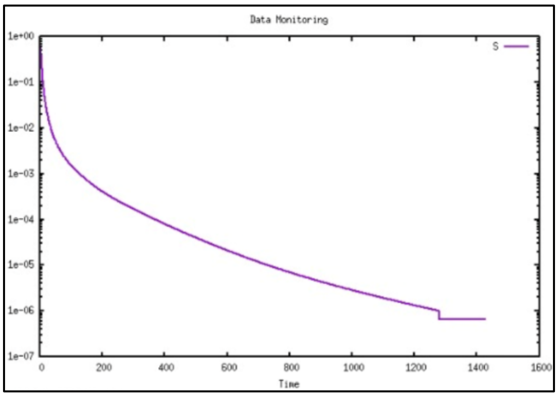


Figure 10 Residue plot for residence time simulations

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 the training algorithm. MLP (Multilayer Perceptron) model was used in training. Geometric coordinates (X and Z) and maximum air residence values 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), and R2 error were used as a loss function for the prediction of the ML algorithm.

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

For Fig. 11 Accuracy ~ 86%, Normalised root mean square error ~ 15 %; For Fig. 12 Accuracy ~ 93%, Normalised root mean square error ~ 10 %

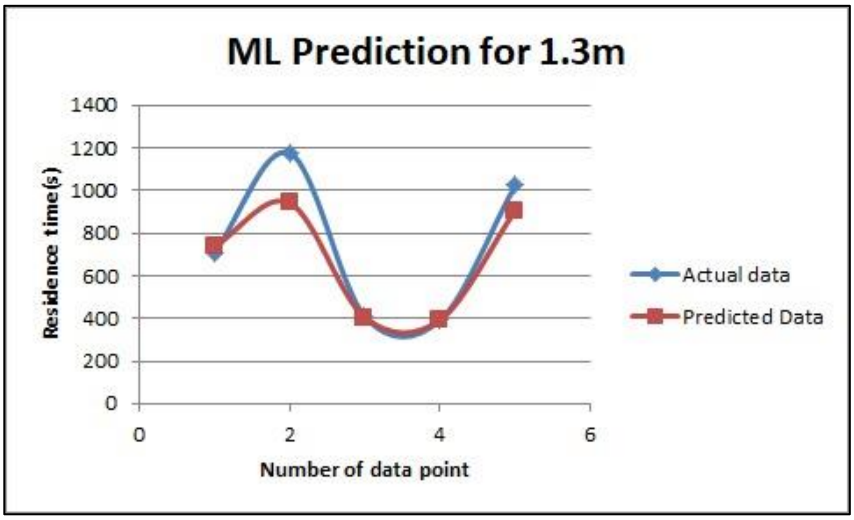


Figure 11 ML Prediction comparison with simulations for 1.3 m plane

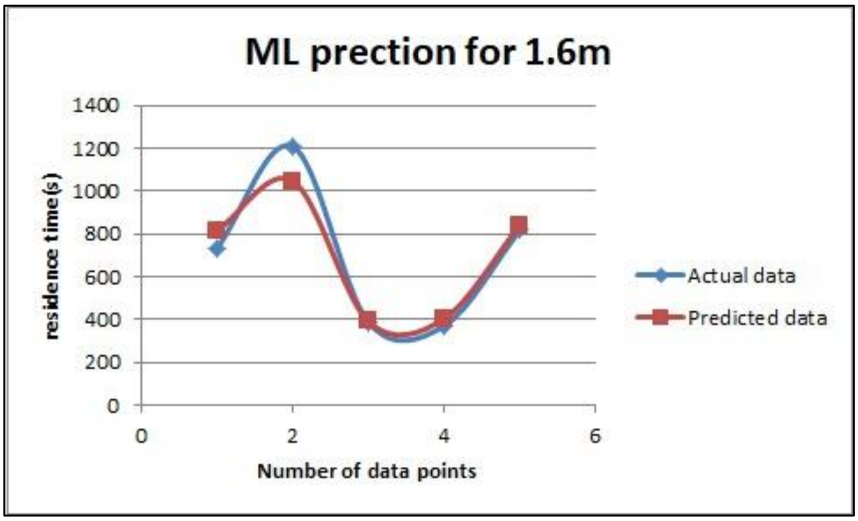


Figure 12 ML Prediction comparison with simulations for 1.6 m plane

- ## For Unique Case

Fig 13(a) and Fig13 (c) show velocity vectors at horizontal planes of 1.3 m and 1.6 m respectively. It can be observed that flow from AC is being merged and taking a little right turn and forming a recirculation zone in the middle, Flow from window AC can completely infiltrate up to the next wall. A smaller recirculation zone is observed at the top right corner of the room. In the bottom corner, the flow is going upward. Similar results are obtained for the horizontal plane of 1.6 m.

Fig 13(b) and Fig(d) show air residence contour at horizontal plane of 1.3 m and 1.6 m respectively. We got a maximum 426-second air residence time at 1.3 m and 411 seconds at 1.6 m. This indicated that the higher height plane is more experiencing suction from the exhaust. Maximum air residence time is observed at the recirculation region.

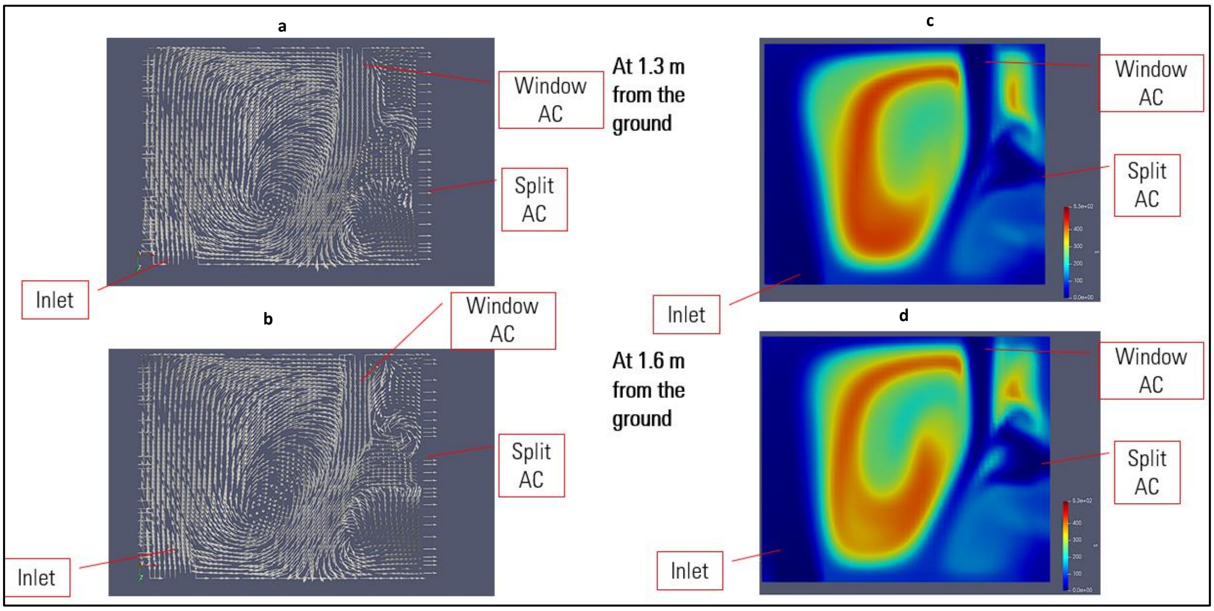
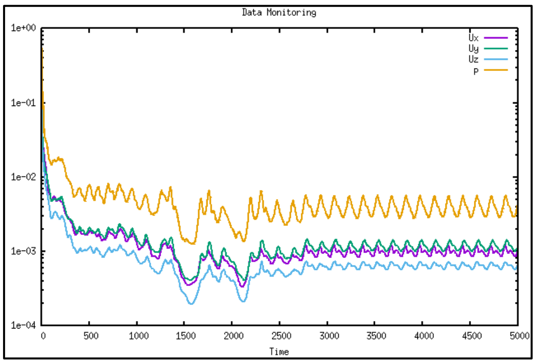


Figure 13 Air residence contours at horizontal top views at a height of 1.3 m from the floor for the unique case with window AC



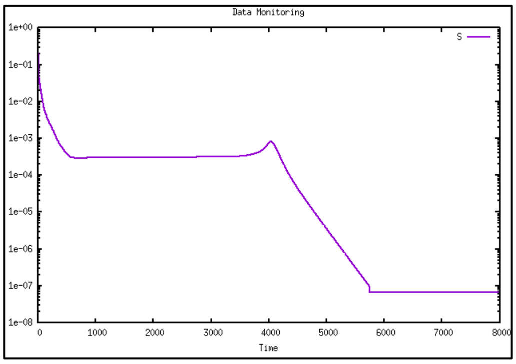


Figure 14 Residue plot for residence time simulations

Figure 15 Residue plot for velocity and Pressure simulation

Conclusion

- In Fig 14, In the front wall, as the location of the exhaust shifts towards the right side, initially air residence time decreases and then again increases this might be due to inlet flow being opposed by AC flow and as the exhaust position is shifted towards the AC, inlet from AC is being drawn by the exhaust.
- In Fig 15, Close to the AC on the right wall, the air stays around for a longer time compared to the area near the exhaust, which is further away from the AC unit. This happens because the airflow is pulled towards the exhaust system.
- In Fig 16, On the left wall, air residence time is significantly higher, this may be because air entering from the door is taken away by the exhaust.

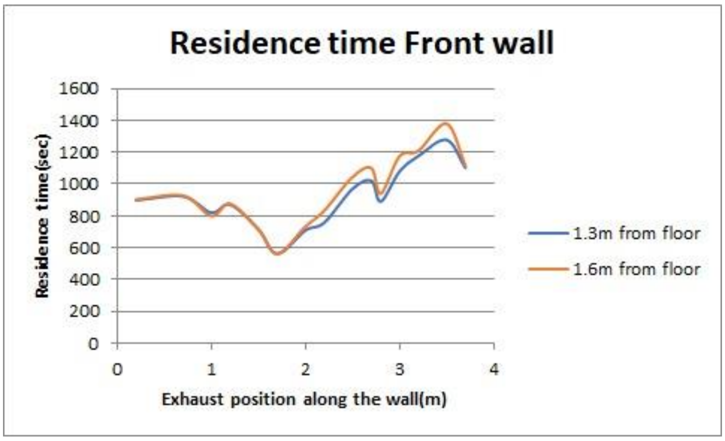


Figure 16 Residence time variations with exhaust position in the front wall

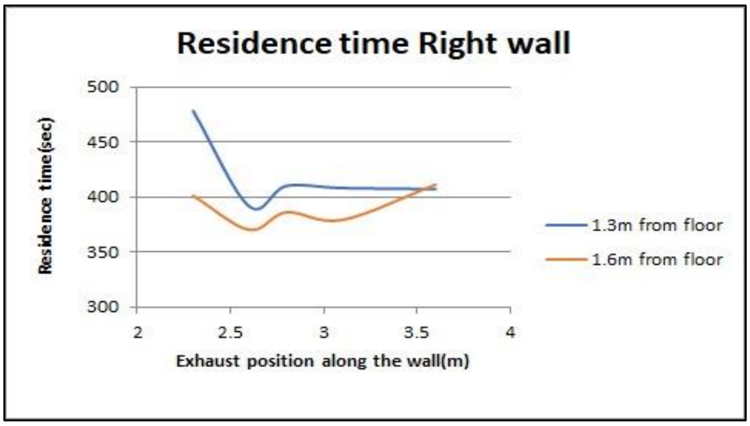


Figure 17 Residence time variations with exhaust position in the right wall

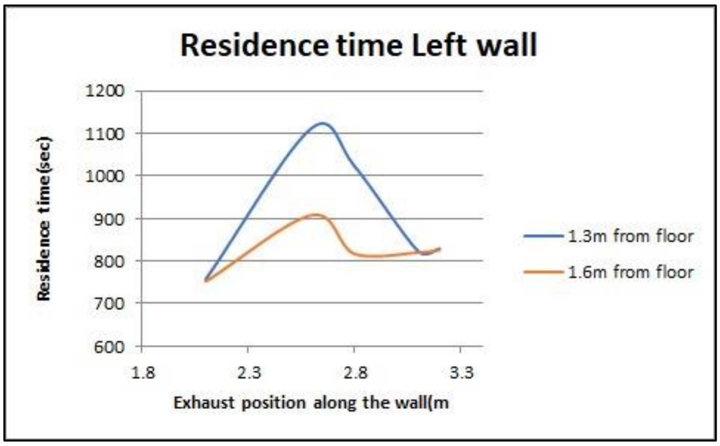


Figure 18 Residence time variations with exhaust position in the left wall

- Average residence time in our common case is about 200 sec. ACH was coming about 18.19 hr^{-1}.
- Average residence time in unique cases is about 167 sec. ACH was coming about 21.44 hr^{-1}.
- ML algorithm can be used to predict the maximum air residence time accurately.
- We recommend using window AC and split AC in the room to reduce maximum air residence time and increase air ventilation rate. This will also reduce the possibility of getting infected with airborne disease.
- We also recommend using high-power exhaust instead of low-power exhaust, as it can withdraw more air and reduce air residence time further.

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).