Airflow management in enclosed spaces using CFD

AE798 M. Tech Project Stage II Report
by

Utkarsh Verma
(Roll No. 193010010)

Department of Aerospace Engineering
Indian Institute of Technology Bombay
Mumbai 400076 (India)

25 June 2021

Supervisor:
Dr. Krishnendu Sinha

Certificate

Department of Aerospace Engineering
Indian Institute of Technology, Bombay

Certified that I approve this M. Tech Phase 2 Project Report titled **“Airflow management in enclosed spaces using CFD”** by **Utkarsh Verma (Roll No. 193010010)** for submission. Certified further that the report represents work carried out by the student to the best of my knowledge.

Date: 25 June 2021

Dr. Krishnendu Sinha

Declaration

At this moment, I, the undersigned, declare that I am the sole author of this thesis. To the best of my knowledge, this thesis contains no material previously published by any other person except where due acknowledgement has been made, and relevant sources have been cited.

Date: 25 June 2021

Utkarsh Verma

Acknowledgments

I wish to express my sincere gratitude to my guide, Professor Krishnendu Sinha. I also would like to feel my gratitude towards Prof. Janani M. from Mechanical Engineering Department and Prof. Guruswamy from Chemical engineering department for helping me in my project. I would like to thank Jagadish Babu Vemula, Pratik, Harsha, Pranav, Subhajit, Pankaj, Mani Shankar for helping me throughout my MTP from the HCFD Lab, IIT Bombay. I also would like to thank Vivek Kumar from Ansys, which helped me throughout phase 1 & 2 of the MTP. It is because of their valuable support and inputs that I can present this report today.

Abstract

Air-borne transmission can pose a major risk of infection spread in enclosed spaces. Venting the air out using exhaust fans and ducts is a common approach to mitigate the risk. In this work, we study the airflow set up by an exhaust fan in a typical shared washroom that can be a potential hot spot for COVID-19 transmission. Primary focus is on the regions of recirculating flow that can significantly increase the residence time of the air. Computational fluid dynamics (CFD) study is performed to get the flow results. In this work, we used the ANSYS FLUENT CFD simulation to study the flow inside the washroom. After getting the flow results from steady state CFD, particle tracking is also performed in order to get the residence time of the actual particles. We identified the large recirculation zone in the washroom, which can be termed as secondary flow or dead zones. This secondary flow harbours a greater number of particles for long time which eventually increase the chances of infection. The effect of exhaust fan velocity is also studied with different CFM of the exhaust fan. To mitigate this secondary flow, we have done some partially opened door to deflect the flow towards the secondary flow zone. By this we can replace the air from the secondary flow zone.

Table of Contents

List of Figures

List of Tables

Chapter 1

Introduction

1.1 Significance of Study

Recently COVID-19 pandemic shock the entire world with the quick transmission of deadly virus from one infected person to another person. The transmission of the COVID-19 also occurs primarily due to the SARS CoV2-laden particles and aerosol particles. The airflow inside any enclosed space is depend on the ventilation system provided there. In general, the fundamental equations of fluid mechanics govern airflow inside the enclosed area like washroom, classroom, hospitals and office space etc. So, the study of the flow physics inside any closed vicinity becomes significant to mitigate the infection risk. This study can also help in creating negative and positive pressure rooms to isolate the patients in hospitals. The flow inside a train, the flow pattern in the critical buildings, offices, classrooms are becoming an important area to study.

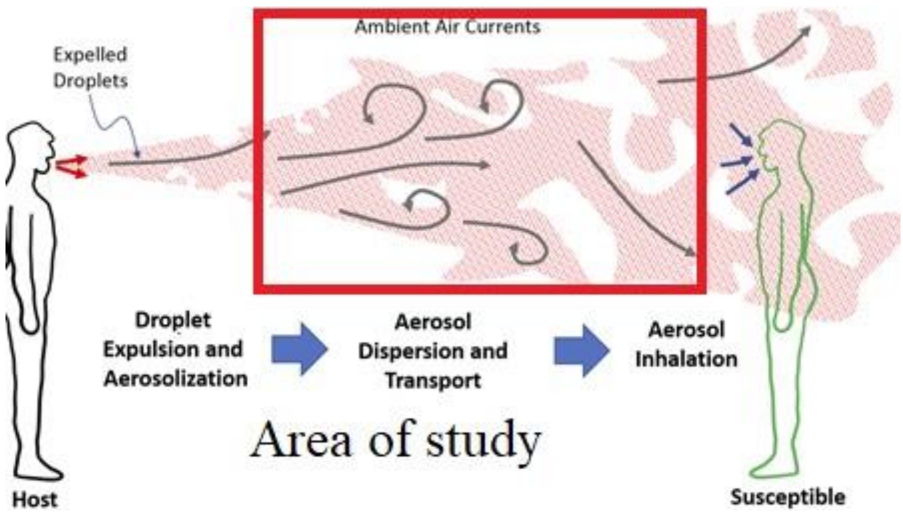


Figure 1. 1 Transportation of aerosol from one person to another

The aerosol particle exit from a particular area depends upon the size of the particle, ventilation rate, flow field, and flow direction. This study has done many CFD simulations in 2 dimensional and 3 dimensional domains by varying the ventilation rate and flow direction. We have found out that the direction of the flow becomes an important tool to mitigate the risk of infection.

Introduction

Aerosol transport within a control volume is primarily affected by the inertial forces due to airflow, drag and lift force on the particles. The computational fluid dynamics (CFD) can help to predict the flow field. These flow fields will help in knowing the potential risky zones inside any area or space. The flow field can be generated in any CFD software (like Ansys Fluent in our case) by numerically solving the Reynolds-Averaged Naiver Stokes (RANS) equation. Different boundary conditions can be given to know the flow pattern.

1.2 Geometry

The washroom of any public area like hospitals, malls, office complex is one of the potential areas where the virus can spread. So, we have done a computational fluid dynamics study for the typical washroom of the size 5ft x 9ft x 10ft. The washroom becomes an important area where the spread can occur because it is used by multiple people, multiple times. The area near the Washbasin in the typical washroom was found to be the most risk-prone area. In this study, the analysis is done on the 2D and 3D geometry of the washroom to mitigate the infection risk, which is explained below.

1.2.1 2D geometry

The typical geometry of the 2-dimensional washroom is given in Figure 1.2. The size of the room is 9ft in length and 5ft in width. The door is 2ft in width, which is located 0.5 ft from the left wall. The exhaust fan for ventilation purpose is located in the middle of the back wall. The size of the exhaust fan is 0.82 ft (250 mm). All the walls are of zero thickness.

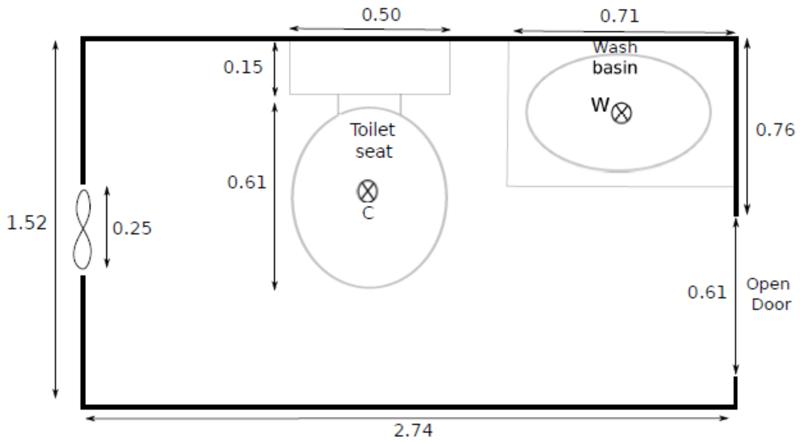


Figure 1. 2 Two-dimensional geometry of the washroom.

1.2.2 3D geometry

We have also verified the 2-dimensional data with the CFD analysis of the 3-dimensional geometry. The 3-dimensional geometry shows the actual flow pattern of the flow inside the washroom. The geometry is with the same plan area, 9ft x 5ft, but the height of the roof is 10 ft. In the 3-dimensional geometry we have also considered the actual fittings inside the washroom like washbasin and the toilet seat. The respective dimensions of the geometry and fittings are given in the figure-1.3. The location of the door is the same as 2-dimensional geometry, but the height of the door is 7ft.

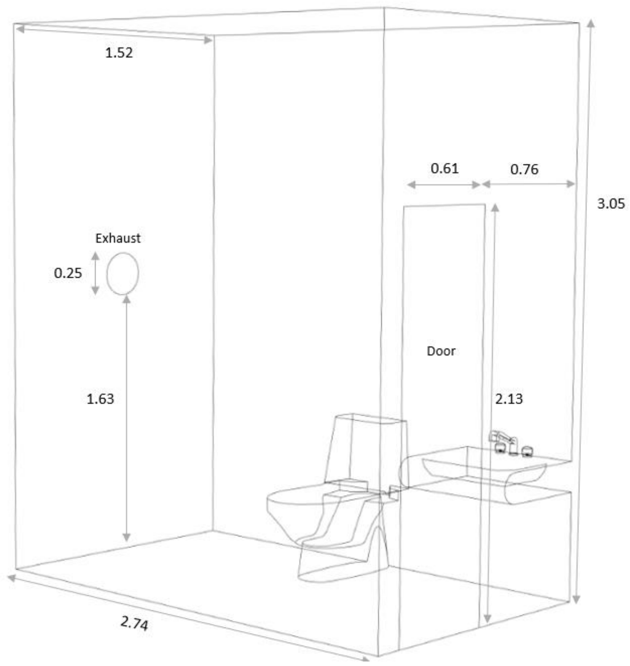
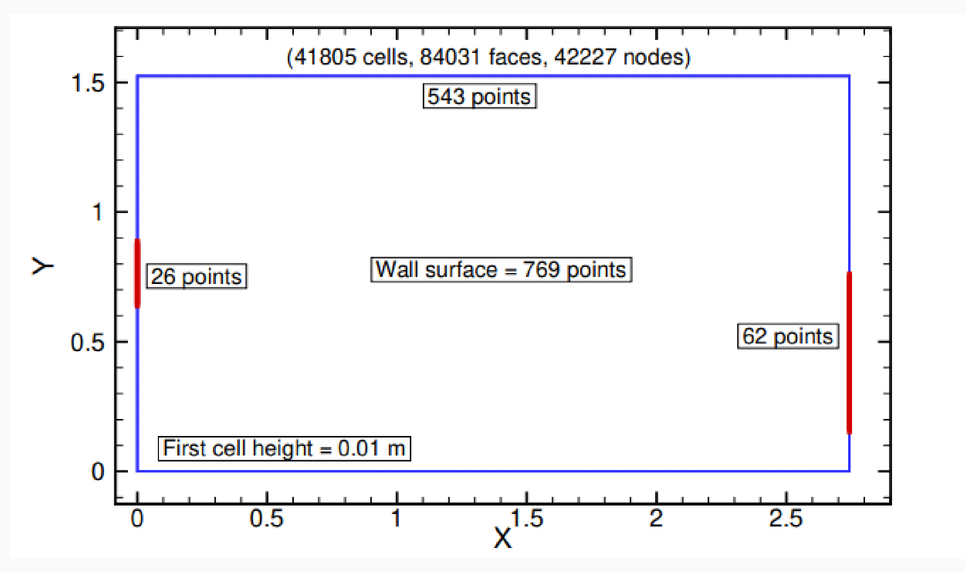


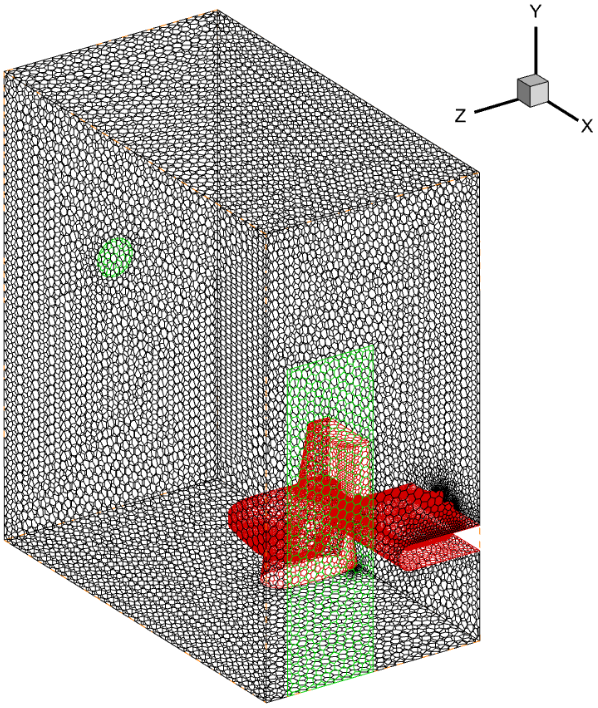
Figure 1. 3 Three- dimensional geometry of the washroom

1.3 Mesh and Grid Details

An unstructured polyhedron mesh is used, as shown in figure 1.4 (a). The mesh was generated using the Ansys meshing software. The different sizes of the mesh size 0.5 cm, 1cm, 2cm, 3cm were considered for the mesh refinement study. For the 1 cm mesh element size, the details of the number of points are given in figure 1.4 (a). The three-dimensional mesh is done with element size of 5 cm which is represented in figure 1.4 (b).



(a)



(b)

Figure 1. 4 Mesh for the washroom in two (a) and three (b) dimensional cases

Chapter 2

Literature Review

There are only a few computational fluid dynamics that have been done primarily related to the COVID-19. Although many studies were done on the ventilation of the air inside the train, buildings, etc.

2.1Flow management inside the classroom model (Abuhgazy, 2020)

Abuhegazy et al. used a 3-dimensional model of a classroom consisting of 9 students and an instructor. The model uses realistic classroom dimensions and air conditioning. The classroom shown in figure 2.1 is 9 x 9 m^2 in area and 3 m in height. The distance between the students is 2.4 m.

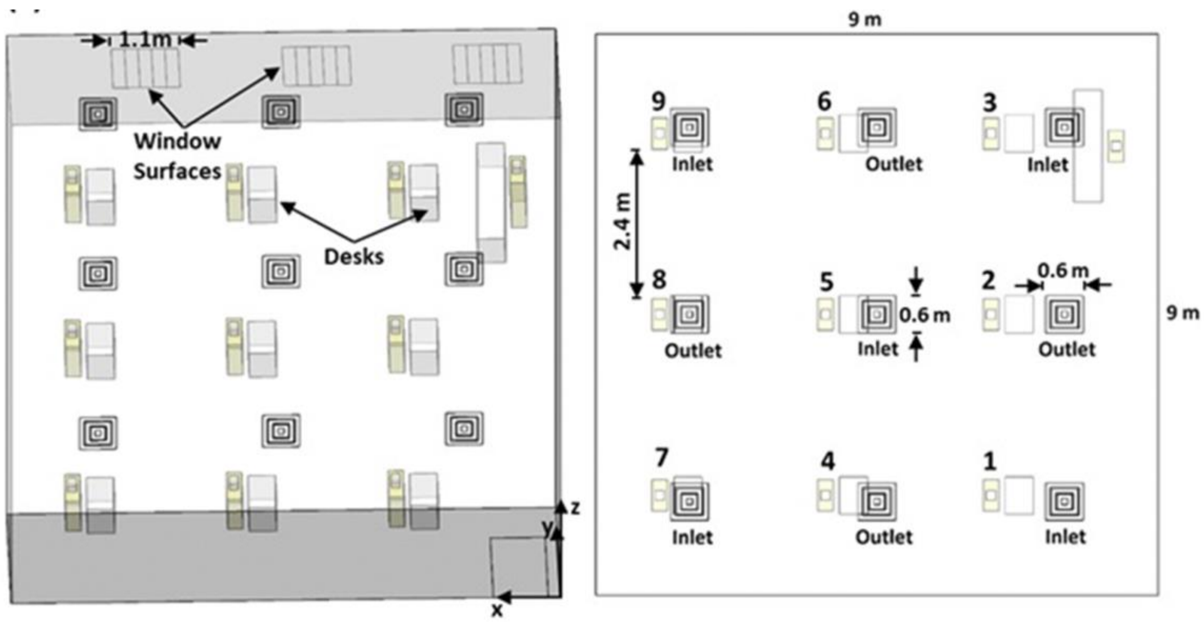


Figure 2. 1 The classroom model (Abuhgazy, 2020)

The above study used the commercial CFD code, ANSYS FLUENT 19.2, to simulate the airflow and particle transport. The mesh which is used is unstructured tetrahedral. The mesh consists of 3.3 x 10^6 cells with a minimum size of 0.5 cm and a maximum size of 10 cm with a gradual transition. They have used the Re- Normalization group (RNG) k- ε model. In this analysis, the governing equations' convection and viscous terms were discretized utilizing the second-order discretization scheme. This study has incorporated the effect of the glass barriers/screens provided to restrict the infection.

In this particular study, to analyze the spread of flu aerosol particles, they considered two conditions. In the first condition, they considered that student 5 spreads the flu. In the second condition, they considered the student eight is spreading the virus. Figure 2.2 (a) shows the spread of flu aerosols spread by student five after 1, 10, 20, 50, 100, and 300 seconds. Similarly, figure 2.1.2 (b) shows the flu particle spread by student eight after the times given above.

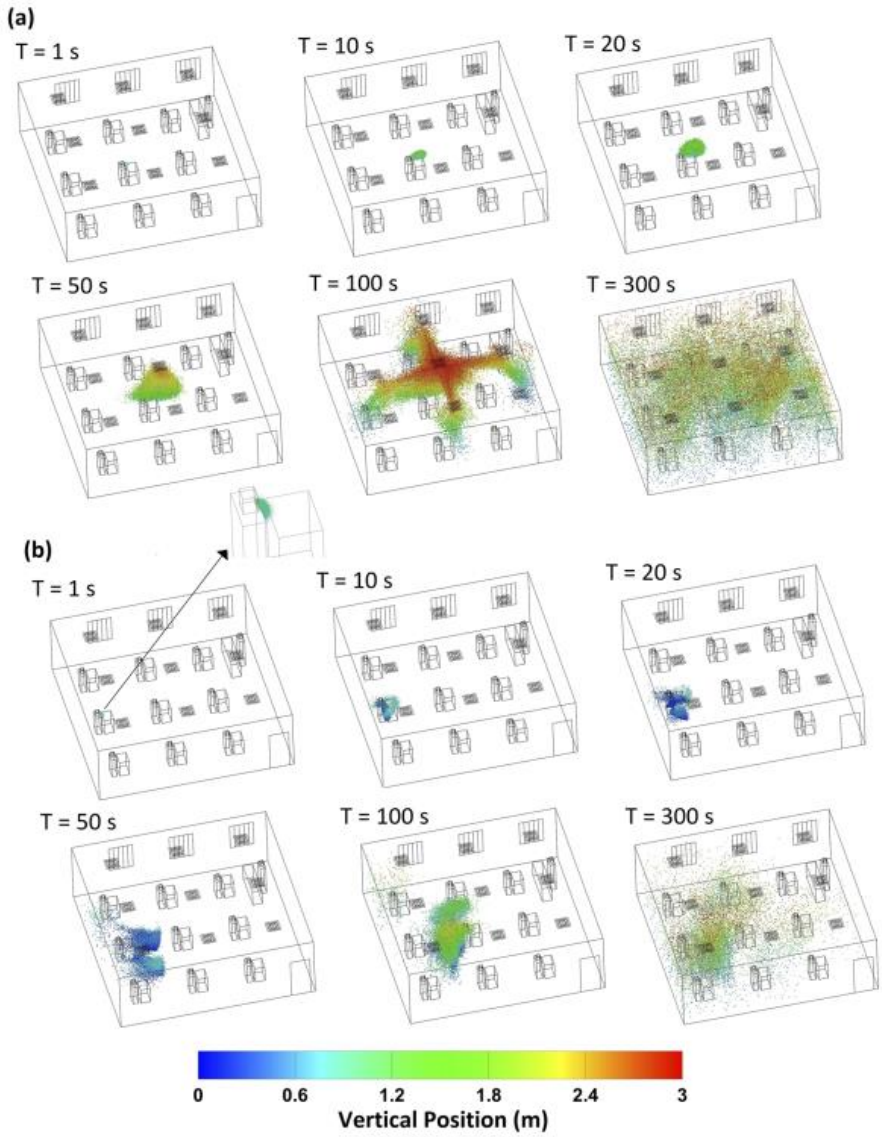


Figure 2. 2 Spread of aerosol particles in the classroom (Abuhgazy, 2020)

To conclude, the aerosol distribution in the room is not uniform, and the air conditioning layout strongly influences it. The effect of source location on aerosol transport is significant. Opening windows while the air conditioning system is running, while not recommended from an HVAC point of view, significantly increases particle exit fraction by ~38% and reduces transmission between students by ~80%. Glass screens reduce aerosol transmission from one student to another and should be used. The extent of their effectiveness depends on the source location for the air conditioning system. Particles disperse in the room and re-concentrate at the return ducts of the air conditioning system. A large fraction of exhaled particles ends up in the air conditioning system, which highlights the need for effective filtration and sterilization systems within air conditioners.

2.2 Flow management inside train compartment (Li, 2012)

Similar kind of studies are done inside the train compartment in Zhang et al. [1] and Wang et al. [2]. To study how the coughed particles disperse in a High-speed railway cabin, they considered the realistic cabin geometry and ventilation system. The size of the cabin is 16 m x 2.85 m x 2.25 m contains 48 passengers in 12 rows. They considered one coughing individual at the G2 location represented in figure 2.3.

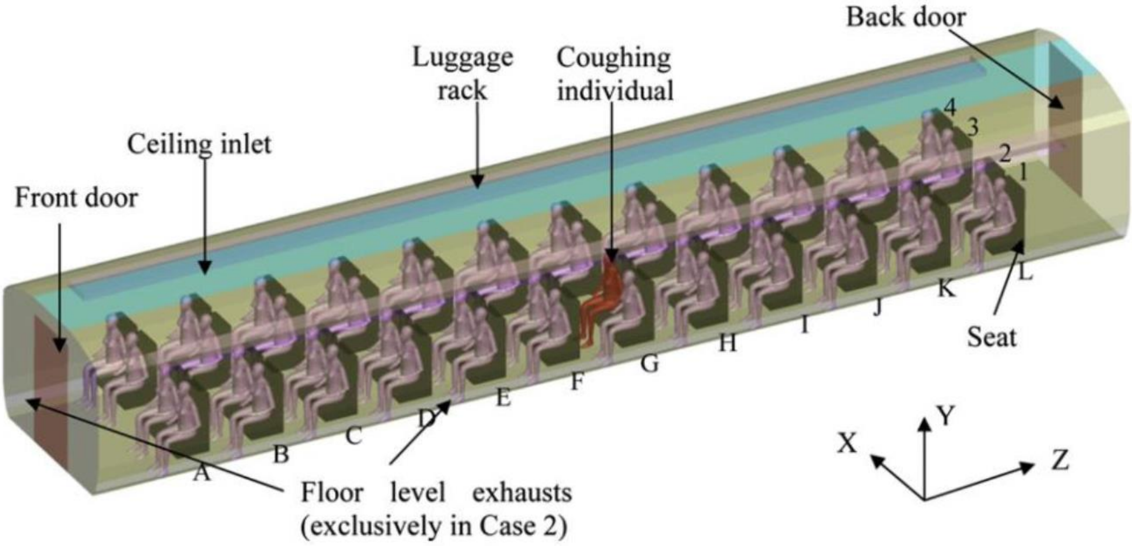


Figure 2. 3 Infected person in train compartment (Li, 2012)

For the flow visualization, they have used the Reynolds Averaged Navier Stokes (RANS) equation. They used different function for the movement of the particle inside the flow. The function of the movement of the particle is given below.

$$\frac{\mathrm{d}u_{\mathrm{p}}}{\mathrm{d}t} = f_{\mathrm{D}}(u - u_{\mathrm{p}}) + \frac{g_{x}(\rho_{\mathrm{p}} - \rho)}{\rho_{\mathrm{p}}} + F_{\mathrm{a}} \tag{1}$$

The four boundary conditions for the ventilation inside the trains are given below.

a. Both the front door and back door were open for exhaust, and there is no through flow.

b. The second condition is such that both the front and back doors were open for exhaust, and one other exhaust is located at the bottom portion of the doors.

c. The third condition is with the through flow from the front doors, and the back doors are used as exhaust.

d. The fourth condition is without the through flow, and the upper roof is used as an exhaust.

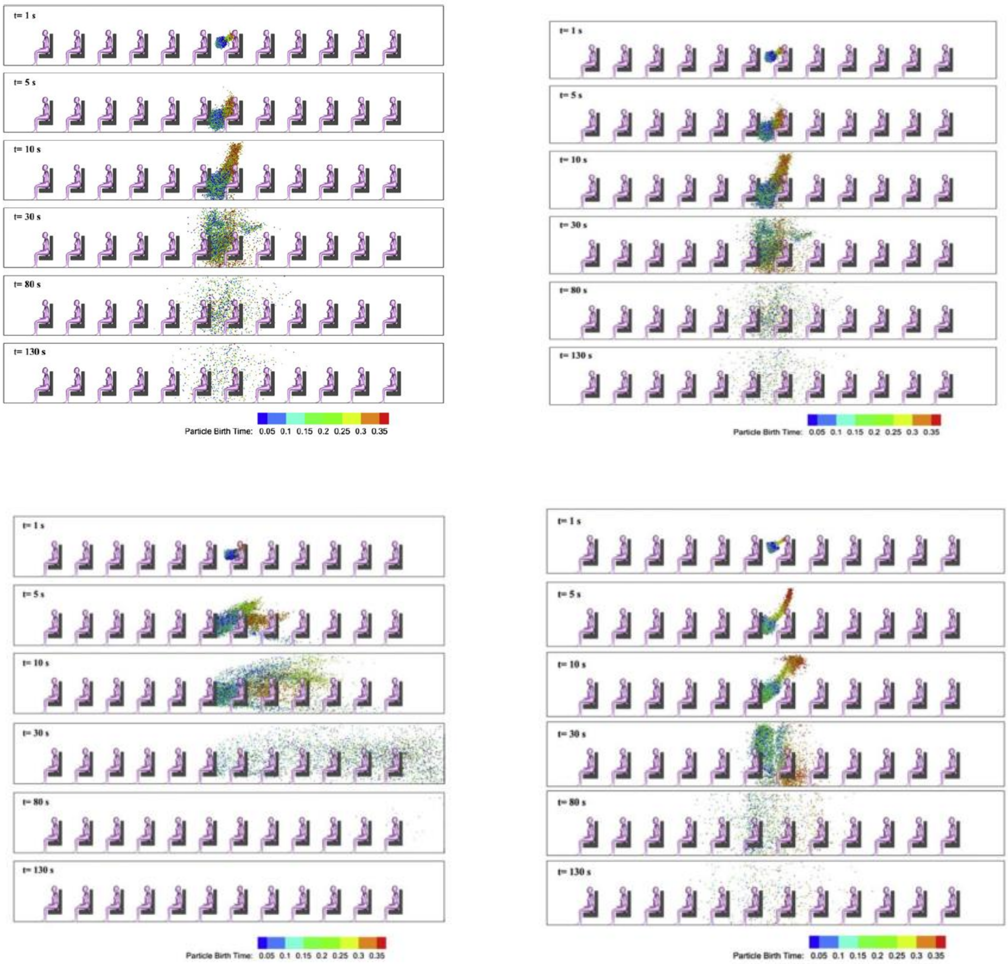


Figure 2. 4 Particle dispersion in different boundary condition (Li, 2012)

In the above study, they found out that in case 3, particle removal efficiency is much higher, but the distance travel by the particle is more extensive than in other cases. Case 4 efficiency is much better than case 2 because the relatively less particle remains in the cabin.

2.3 Flow management inside theater (George, 2016)

A similar kind of study is also done in the socio-cultural building in Romania. Here they have not incorporated any aerosol particles. They just compared the ventilation system (HVAC) efficiency during the spectacles hours and the breaks. The longitudinal section and the cross-section of the building are given in figure 2.5. The velocity contours are given in Figure 2.6 (a) and (b).

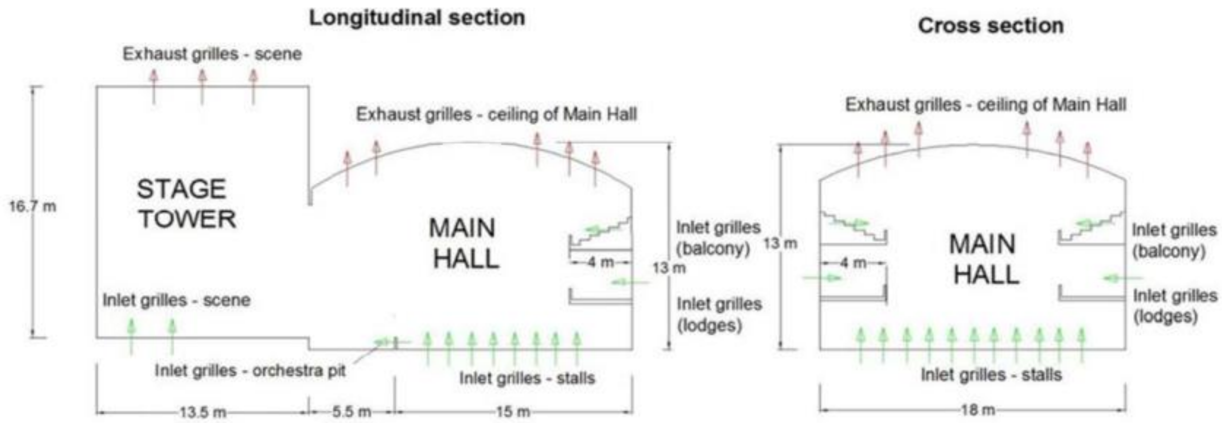


Figure 2. 5 Longitudinal section and cross-section of the building. (George, 2016)

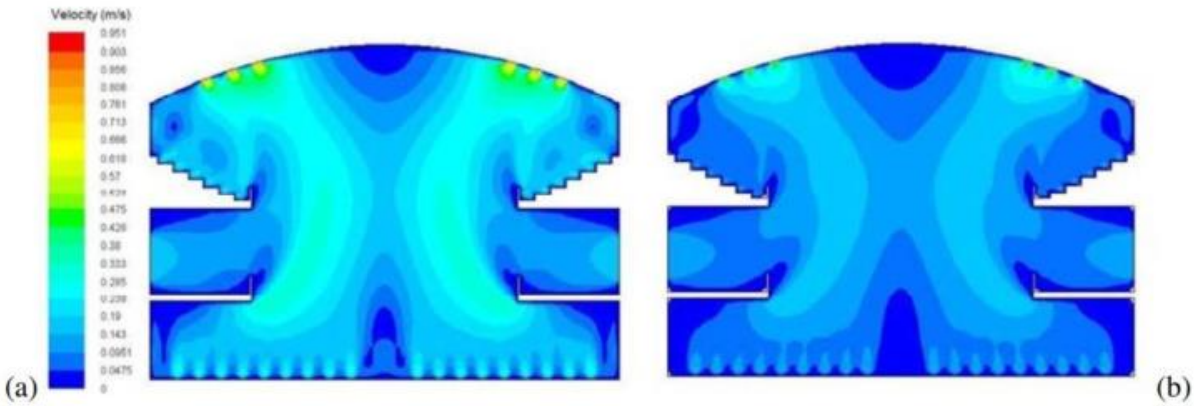


Figure 2. 6 velocity contours (a) during spectacles; (b) during breaks (George, 2016)

They have found out that the load on the HVAC system is much more than when the hall is empty during the spectacles.

2.4 Simulations inside dental clinic (Peyvan, 2021)

A similar kind of recent study was done in a dental clinic by Jonathan et.al. In this study the evaporating droplets were tracked inside the dental clinic during the ultrasonic scaling procedures. The simulation also considers the 25 patients treatment cubicles in an open plan dentistry clinic. The model of this clinic is given in the figure 2.7. In the layout of the clinic red and blue circle denotes the ventilation outlets and inlets respectively.

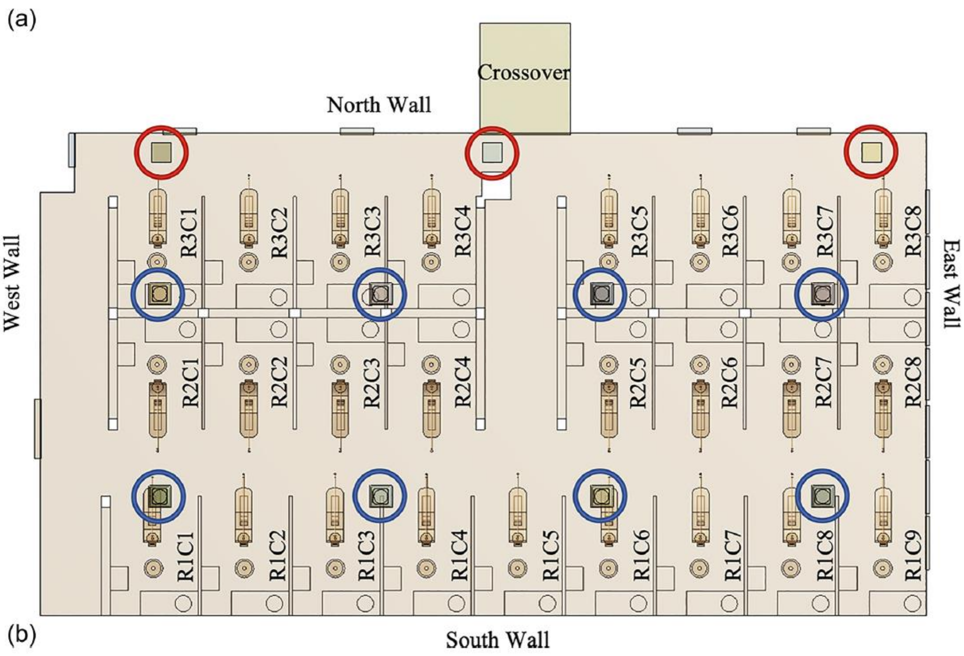


Figure 2. 7 Layout of the dentistry room (Peyvan, 2021)

In this study the transportation of droplets and aerosols generated is determined by the discrete phase model in ANSYS FLUENT 2020R1. (Peyvan, 2021) This study also includes the evaporation model in their simulations. There are various conclusions were drawn from this study that the location of the aerosol in the given space will largely depend on the relative position of the duct. The different ventilation rate vent can create turbulence inside the room. Spread will create a disparity in the behavior of droplets in different area. The smaller size of the aerosol will remain for more amount of time. They have given that for droplet diameter less than 15 micron will stay more than 7.13 min. So, this study creates a huge impact in terms of the residence time of the particles. If the particles will stay for long then there will be high chances of infection. Figure 2.8 shows the velocity streamlines inside the domain. They are colored by the originating vent.

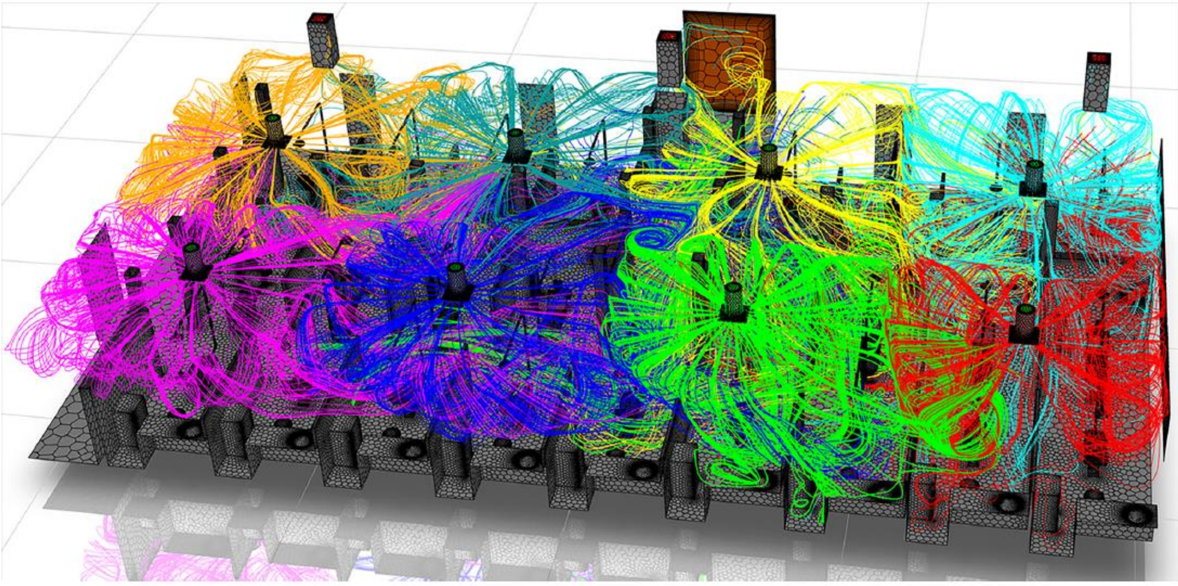


Figure 2. 8 Velocity streamlines originating from vent (Peyvan, 2021)

There are some more studies in this area, and the recent COVID-19 motivated us to do the same kind of study. We have done the same kind of study inside the washroom, which is explained in the next chapter.

Chapter 3

Simulation Methodology

3.1 Governing Equations

The flow is governed by principles of mass conservation, momentum conservation, and energy conservation. These physical laws come together in a set of equations –

$$\frac{\partial \rho}{\partial t} + \frac{\partial(\rho u_i)}{\partial x_i} = 0 \tag{3.1}$$

$$\frac{\partial}{\partial t}(\rho u_i) + \frac{\partial}{\partial x_j}\left(\rho u_i \cdot u_j\right) = \rho g_i + \frac{\partial \tau_{jj}}{\partial x_j} \tag{3.2}$$

$$\frac{\rho DE}{Dt} = \rho g_i u_i + \frac{\partial(\tau_{ij})}{\partial x_j j} - \frac{\partial q_j}{\partial x_j} \tag{3.3}$$

Where, $\frac{\partial q_j}{\partial x_j}$ represents the conduction flux

Were,

t = Time
ρ = Density
p = Pressure

x_i = i^{th} direction length
u_i = Velocity in i^{th} direction
u_j = Velocity in i^{th} direction
q_i = Heat flux
u_i' = velocity fluctuations
E = Energy

τ_{ij} = Element of stress tensor obtained by Stokes's hypothesis

Equations 3.1, 3.2, and 3.2 represent the mass, momentum, and energy conservation for the infinitesimal volume. They are the Navier stokes equations written in the conservation form.

The Reynold's Average Navier Stokes Equations- The N-S equations are difficult to solve exactly, and hence we use a system of time-averaged equations, which are the Reynold's Average Navier Stokes equations. Further, the 3D conservation form of the system of equations can be written as-.

$$\frac{\partial u_i}{\partial t} + \bar{u}_i \frac{\partial u_i}{\partial x_j} = -\frac{\partial \bar{p}}{\partial x_i} + \partial \frac{\sigma_{ij}}{\partial x_j} + \frac{\partial}{\partial x_j}\left(\overline{-u_i' u_j'}\right) \tag{3.4}$$

Where, u_i', u_j' represents the fluctuations in the velocity and the $\bar{u}$ represents the mean velocity. The term $\frac{\partial}{\partial x_j}\left(\overline{-u_i'u_j'}\right)$ represents the Reynolds stress.

3.2 Numerical Methods

ANSYS FLUENT used the finite volume approach for solving the RANS equation. The exact solution of the RANS is difficult to get, so We discretize them into system of algebraic sum. The finite volume approach is one of the methods which is used in the discretization.

3.3.1 Finite Volume Formulation

In the finite volume method, the entire domain is divided in to sub-domains. These sub-domains are called as control volume. After dividing the into the control volume, we need to integrate the governing differential equation over each control volume. Now to simplify the process of formulation the system is reduced to two-dimensional system. Now the conservation equation 3.4 is integrated over the control volume and control surface to give:

$$\frac{\partial}{\partial t}\iiint_{cV} U\, dV\ t \iint_{S} \left[\left(\overrightarrow{F_1}+\overrightarrow{G_1}\right)+\ \left(\overrightarrow{F_V}+\overrightarrow{G_V}\right)\right].(\hat{n}dS) = Q$$

Here $\hat{n}$ represents the unit vector normal to the surface are dS. U represents the set of conservative variables. $\overrightarrow{F}$ is the flux vector in streamwise direction. $\vec{G}$ is the flux vector in flow normal direction.

3.3 Comparison of Turbulence Models

There are several models developed for studying the turbulent flows. We have two-equation models, such as the standard k-omega model and k-epsilon models. These take in an eddy viscosity hypothesis. The use and significance of the models are briefly understood here.

The k – epsilon model

- equation model solves for the turbulent kinetic energy - k and the rate of dissipation (epsilon) of k

- Good for predicting flow variables far from walls, plane shear layers, and recirculating flows.
- It does not give good results for flows with large pressure gradients. It should not be used for unconfined/rotating flows/curved boundary layer flows.

The k - omega model

- Two equation model solves for the turbulent kinetic energy - k and the specific rate of dissipation (omega) of k into thermal energy.
- Useful for near-wall predictions.
- Solves for turbulent eddy viscosity - $\mu T = \rho k$/omega: The total viscosity is then $\mu total = \mu + \mu_T$
- **Shear Stress Transport (SST) Model**
- • Combines the k - omega and k - epsilon model for good results near as well as far from the wall.

For the present study we have used the realizable k-epsilon model with enhanced wall treatment. Realizable k-epsilon model uses a different method to formulate the turbulent viscosity. This model shows substantial improvement over other standard k-epsilon model where the flow feature includes strong streamline curvatures, rotation and vortices.

3.4 Calculation of the residence time of the air

The residence time is calculated on the principle that the time which air spends in a domain after entering from the inlet and before going out from the exhaust. That means the residence time of a fluid passing through a small space is the age of the air.

Let say a general scaler equation of flow with no diffusion term

$$\frac{\partial \rho_{\emptyset}}{\partial t} + \frac{\partial \rho_{u\emptyset}}{\partial x} = S \qquad (3.5)$$

Assuming the flow to be steady state, then the equation will drop the time term.

$$\frac{\partial \rho_{u\emptyset}}{\partial x} = S \qquad (3.6)$$

$$\frac{\rho u(t+\Delta t)-\rho ut}{\Delta x} = S \tag{3.7}$$

$$\frac{\rho u(\Delta t)}{\Delta x} = S \tag{3.8}$$

But the relationship of the flow rate and the residence time is a small space is given by

$$u = \frac{\Delta x}{\Delta t} \tag{3.9}$$

On putting equation (3.9) in the equation (3.8) will give that

$$\rho = S \tag{3.10}$$

This source scan section on the right-hand side corresponds to the density. So, if we provide the source as a density of the air, will give us the residence time or age of the air.

3.5 Particle dynamics [8]

Once the solution is converged, we have done particle tracking to transport the water liquid particles. Particle dynamics is done to see the behavior of the actual particle in the primary and the secondary flow zones inside the washroom. The behavior of the particles is determined by solving the equation of motion for the particles in Lagrangain framework [8]. The equation 3.7 of the flow is given by

$$m\frac{dv_i}{dt} = \overrightarrow{F_{drag}} + \vec{F_g} + \overrightarrow{F_a} \tag{3.7}$$

where, v_i represents the velocity of the particle, m is the mass of the particle. $\overrightarrow{F_{drag}}$ represents the drag force acting on the particles. In our case, we have used spherical drag model (Peyvan, 2021). The same model is also used by the (Peyvan, 2021). $\overrightarrow{F_a}$ represents the additional forces acting on the particles. The additional forces include virtual mass force, pressure gradient forces, Saffman lift forces etc. But for the particle size 0.5 to 20 micron, the effect of the virtual mass force, pressure force is negligible (Ahmadi, 1992). Therefore, we have included only Saffman lift force. The Saffman lift force (Peyvan, 2021)includes the shear of the particles and this force is best suited for the spherical shape particles. In our case we have included Saffman lift force. $\vec{F_g}$ represents the force due to gravity acting on the particles. The effect of the turbulence and the dispersion of the particles was taken into account by choosing discrete random walk model of the particles.

For the present study we have used the 350000 particles approx. which are sufficient to predict the patten of these particles. The size of these particles is 5 micron which is in the range of the cough particle. The present study has not taken the effect of vaporization of the particles in to account to see how much time a particle can spend in the secondary flow. In the reality there will be vaporization of particles after few seconds only. In the reality particles will also reflect and make films while striking to the wall or any other boundary inside the domain. But for the simplicity we have neglected that effect, and as soon as the particle will strike any boundary or surface it will trap on the wall.

Chapter 4

Details of the simulation

In this chapter we will describe the various details for the simulation which are used.

4.1 Model mesh details

For the present study, we have compared the two different sizes of the mesh, one is 10 cm and other one is 5 cm. The meshing done with the polyhedron cell type. The total number of the grid points are 867119 and the number of elements is 320066. The figure 4.1, which is the plane in the y-direction taken at a height of 1.8 m from the floor of the washroom, shows the mesh refinement study, the flow pattern is almost similar in both the cases. For the further calculation we have used the 5 cm mesh. In which the redline is corresponds to the x-velocity magnitude with 10 cm mesh and green line represents the x-velocity line with respect to 5 cm mesh.

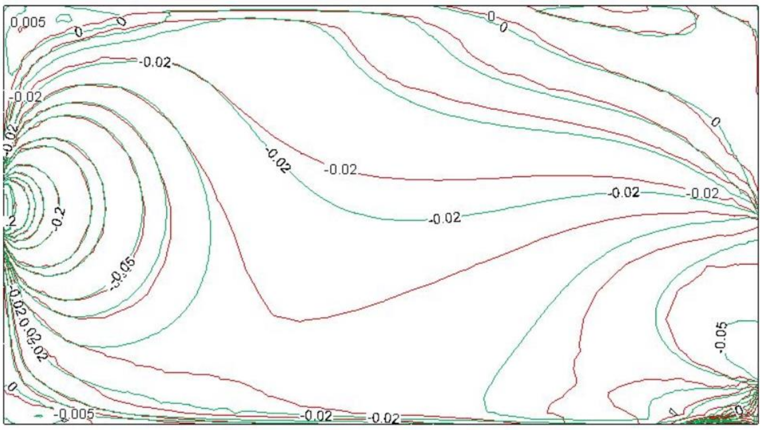


Figure 4. 1 shows the velocity contours lines for two different mesh size in which red lines shows 10 cm grid velocity and green line shows the velocity contour lines with 5 cm grid

4.2 Boundary condition for the steady state simulation

4.2.1 Boundary condition in 2D flow

Table 4. 1Boundary conditions for two-dimensional flow

Inputs	Value
Model	Realizable k-epsilon model

Details of the simulation

Exhaust velocity	0.3 m/s
Inlet	0 Gauge pressure
Turbulent Intensity	5%
Turbulent viscosity ratio	10
Temperature	300 K
Wall Condition	Stationary wall and no-slip
Solver Details	Pressure based, Steady-state, coupled scheme

4.2.2 Boundary condition in 3D flow

Table 4. 2 Boundary conditions for three-dimensional flow

Inputs	Value
Model	Realizable k-epsilon model
Exhaust velocity	1.5 m/s
Inlet	0 Gauge pressure
Turbulent Intensity	5%
Turbulent viscosity ratio	10
Temperature	300 K
Wall Condition	Stationary wall and no-slip
Solver Details	Pressure based, Steady-state, coupled scheme

4.3 Boundary condition for the transient simulation

For the visualization of the particles, we have done the transient simulation to capture the behavior of the particles at different time instances. The details of the particle tracking are given in the Table 4.3.

Table 4. 3 Input parameter for particle tracking

Inputs	Value
Size of the particle	5 microns
Type of particle	Water- liquid
Velocity in x- direction	0.1 m/s
Velocity in y- direction	0.1 m/s

Details of the simulation

Velocity in z- direction	0.1 m/s
Total mass injected	2.85 x 10^{-9} Kg
Time step size	0.005 sec
Total time steps during injection	1000
Height of injection	1 m from floor

4.4 Injections for the particle tracking

The above-mentioned injection is the point injection done at the two different locations. Figure 4.2 represent the two locations of the injection. First injection is at the center of the washbasin which is in the secondary flow and other one is at the center of the washroom which is in the primary flow.

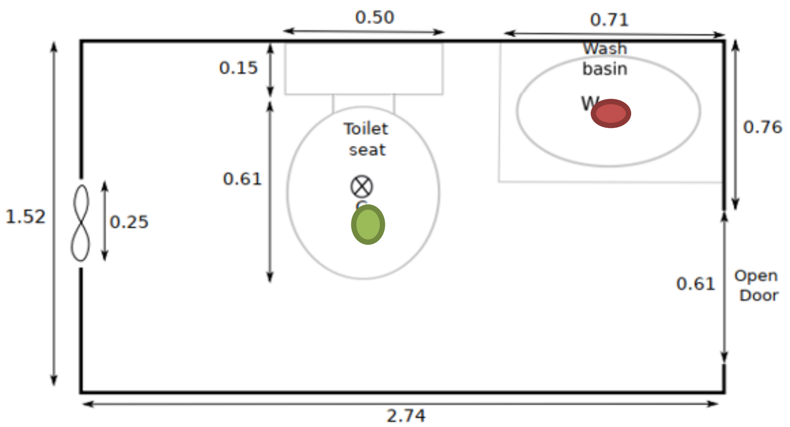


Figure 4. 2 Location of injection (a) Green in Primary flow (b) Red in secondary flow

After the injection, we have tracked the number of particles in the different zones inside the washroom. The zones were taken on the basis of flow pattern. Some zones were taken in the secondary flow and some in the primary flow in order to capture the behavior of the particles. The details of the injection are given in the table 4.5 and the figure 4.3 represents the corresponding location in the 2D view.

Details of the simulation

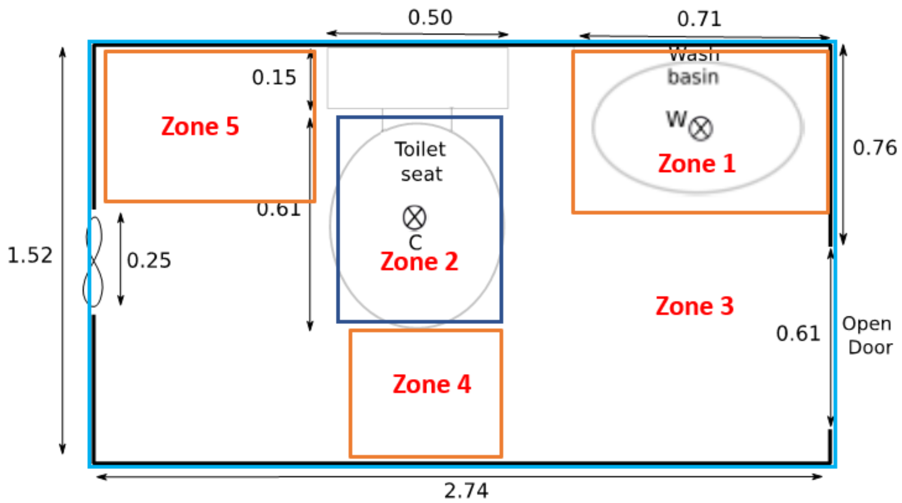


Figure 4. 3 Locations of the zones for particle tracking

Table 4. 4 Details of the Zones

Zone	Specification	Y Location
1	At washbasin	Y= 0.4 m to Y = 2.0 m
2	Centre of the washroom	Y= 0.5 m to Y = 2.0 m
3	Covering entire ceiling	Y= 2.5 m to Y = 3.2 m
4	In primary flow	Y= 0.4 m to Y = 1.5 m
5	Beside toilet seat	Y= 0 m to Y = 0.4 m

Chapter 5

Simulation Results

For the analysis, we have done 2D and compared it with the 3D results. The geometry of the washroom is given in chapter 1. For the two-dimensional analysis, we have not considered the effect of the different fittings provided inside the washroom. But the three-dimensional flow visualization is done with the effect of the fittings like (Washbasin, toilet seat, taps, etc.).

4.2 Two-dimensional flow results

The results of the flow are given in figure 5.1. The flow contains the primary flow and the few places the secondary flow. The secondary flow becomes the concerned area because here, the flow is circulating. These secondary flow zones are termed as the dead zones. If any aerosol particle goes into these dead zones, then it will not come out of the domain. The air will spend more time in this zone and eventually increase the residence time. If any person will sneeze in this area the particles are likely to spend more time than sneezing in primary flow.

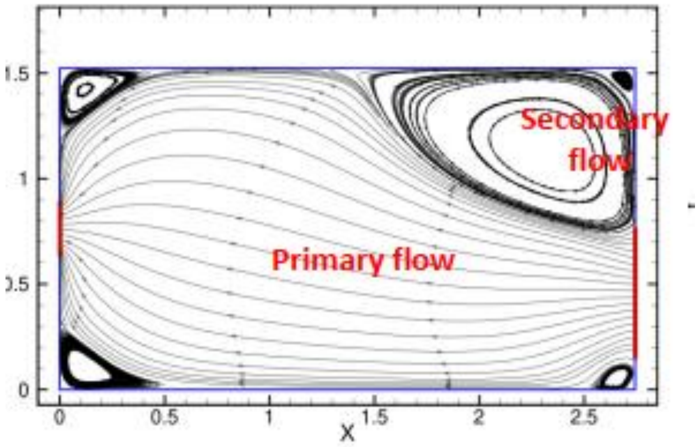


Figure 5. 1 Streamlines inside the washroom in the 2-dimensional simulation

In Figure 5.1, we can see the five dead zones are present. If any fluid particle will come into this zone, it may increase the risk of the infection to the second person. Figure 5.2 shows the velocity magnitude lines. There are points where the velocity is zero. In those areas, there are two possibilities. The first is where the vortex of the secondary flow is present, and the other one where the reattachment point is there.

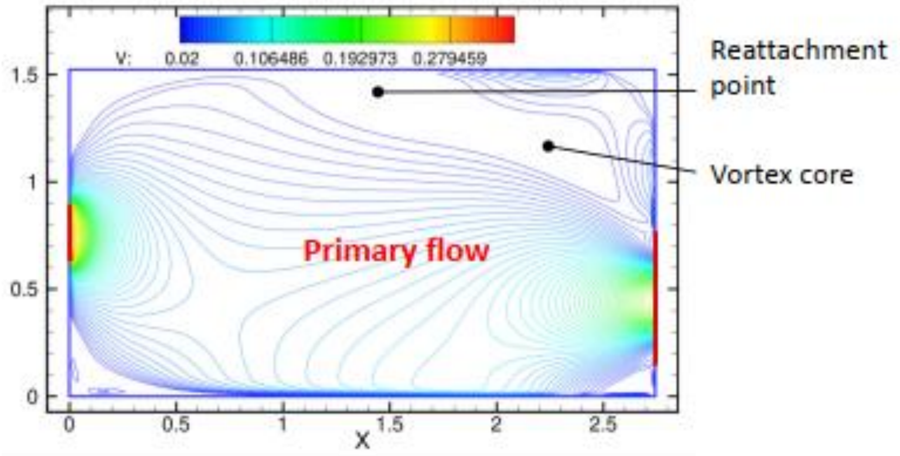


Figure 5. 2 Velocity magnitude lines in the 2-dimensional simulations

But 2D flow is not the real picture for which we are concerned. Hence, we have actual flow simulation in the 3D case.

5.3 Three-dimensional flow results

The 3D simulation shows that the flow in the actual condition is too complex. For the simplicity of the analysis, we have taken several planes on the Y-direction to visualize the results.

Figure 5. 3 Complex flow inside the 3D geometry of the washroom

5.3.1 X- Velocity contours

The figure 5.4 shows the x-velocity contours in the different planes in Y-direction. The area which is of white color shows the reverse flow of velocity. This reverse flow of velocity signifies the presence of the recirculation zone. The plane at the 1.8 m height coincides with the exhaust fan, door. This plane is qualitative representation of the two-dimensional case.

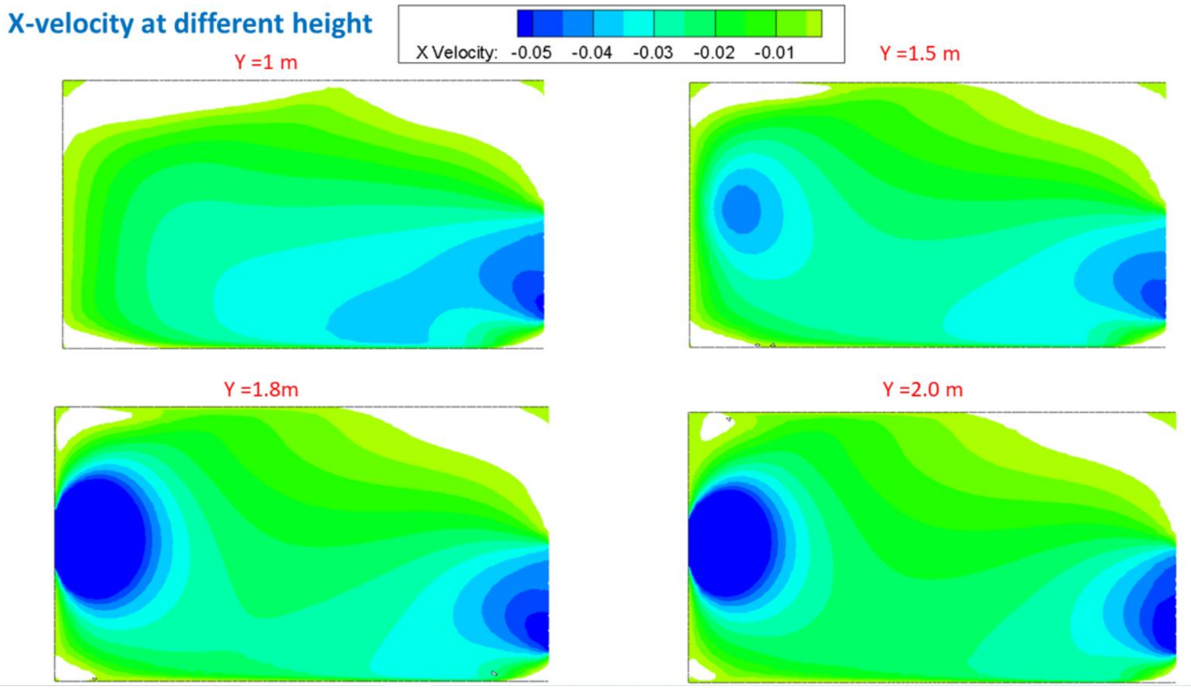


Figure 5. 4 Velocity contours at different horizontal to visualize the complex three-dimensional flow field computed in the washroom

5.3.2 Velocity vectors

To show the recirculation of the flow we have plotted the velocity vectors at different planes which shows are formation of the recirculation zone at different planes in different location. We have taken 4 planes according to the formation of the recirculation zones at the different planes. First case is Y=0.35 m plane, which presents the two recirculation zones one is below the washbasin which is of not much significance. Other one is near the toilet seat which can be significant, if a person is using the toilet seat. To analyze this second recirculation zone at 0.35 m plane, we have taken zone 5 into consideration. We have counted number of particles in different time. Second case is Y= 1.5 m from floor plane. This plane represents the position of human face level. Here we can again observe two recirculation zones. One is very small near the toilet seat and another one is above the washbasin. The recirculation zone above the washbasin is of more concern. For this we have taken Zone-1 in consideration to capture the particles in this area. Another area of concern is Y=3.0 m, which represents the ceiling of the washroom. In this area we can see the almost all the area has the recirculation zone. The velocity vectors have the direction opposite to the flow. The entire ceiling is becoming the sink where the particles are going after some time. This area can be of much significance when the height of ceiling is too low or when we are using the ceiling fan. To analyze the particles in this zone we have taken zone 3 into consideration.

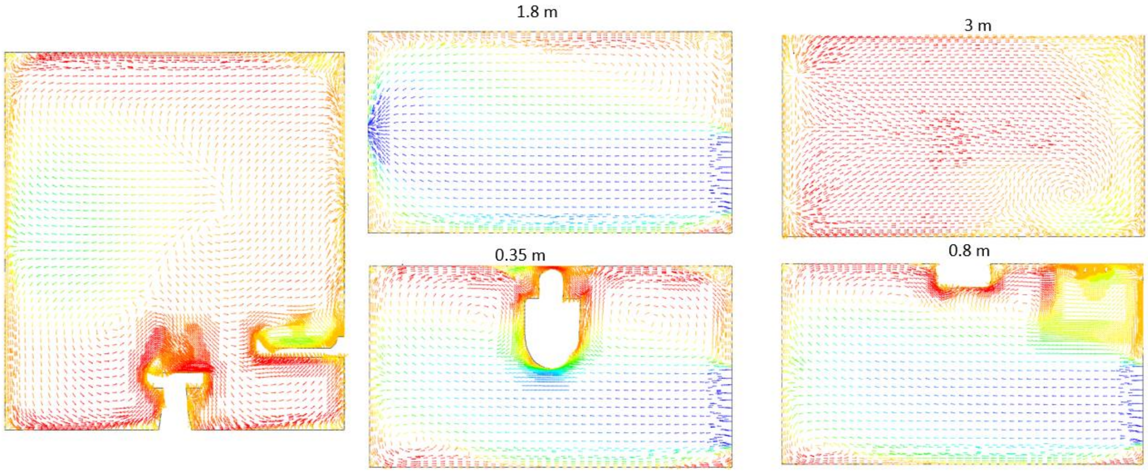


Figure 5. 5 Velocity vectors at different horizontal and vertical planes to visualize the complex three-dimensional flow field computed in the washroom

5.3.3 Residence time

The residence time is essentially the age of the air inside the domain. To visualize the residence time also, we have again taken different planes. On these planes we have analyzed that which area has more residence time and which has less. The primary flow has the residence time of 50 sec and the secondary flow has the residence time more than 1000 seconds. This signifies that the secondary flow zone or dead zone have the residence time 20 times more than the primary flow time. From the figure 5.6, it is clear that the washbasin zone has more residence time than all other zones. So, from all other dead zones which are forming at different location in the washroom the washbasin dead zone is of utmost important. This zone will harbor a greater number of particles.

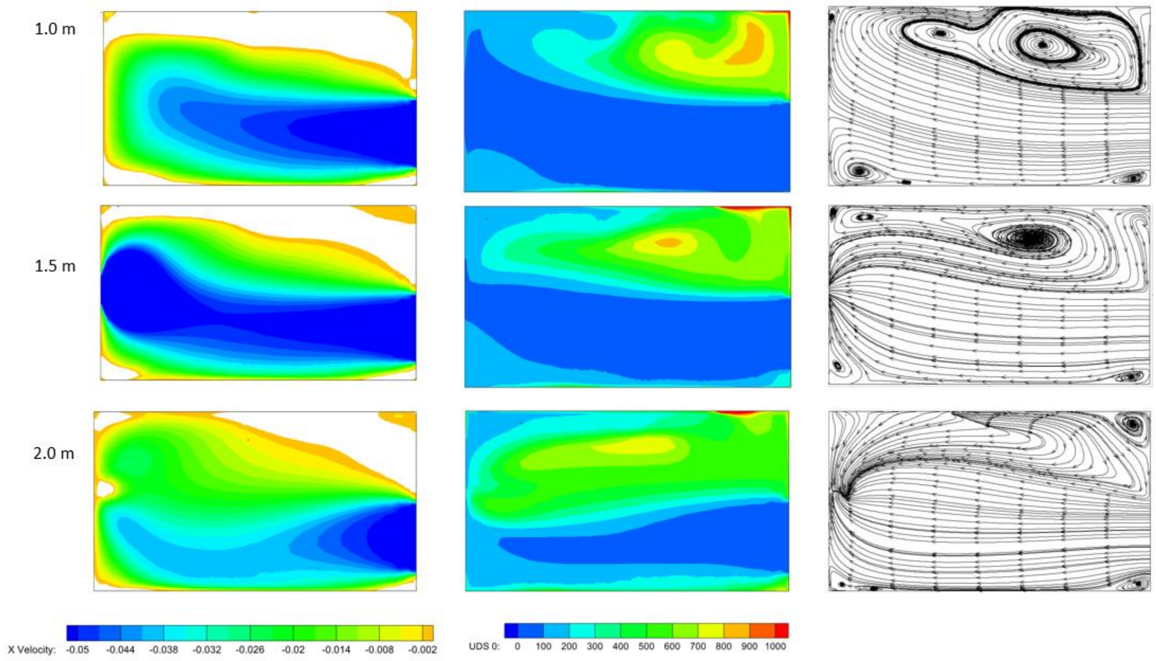


Figure 5. 6 Contours of x-velocity (left), air residence time (middle) and streamlines (right) plotted in different horizontal sections of the washroom.

5.4 Particle Tracking results in three-dimensional flow

The particles are visualized in different zones mentioned above in table 4.3 For the purpose of analysis we have injected the particles at two different locations. The results at two locations are given below. The figure 5.7 and 5.8 shows the top view of the washroom.

5.4.1 Injection at the center of washbasin

Once the flow is converged then the particles are injected at the center of the washbasin for 0.5 sec. The total mass injected at the center of washbasin is 2.85×10^{-9}. Figure 5.7 shows the location of the particles at different times after the injection. The color of particle representing the y- position of particles. The blue color particles means that the particles are at ground level and the red color shows that they are present at ceiling level. After 50 seconds the first particle will go out of the exhaust. At this point the particles will be in the primary and secondary flow both. As the time increases the primary flow will be vacant and only the secondary flow will harbor particles for longer duration. If we see after 300 or later the particles in primary flow are very less. But in the recirculation zone like washbasin area and ceiling harbor more particles. After some the particles in the primary flow will further reduce.

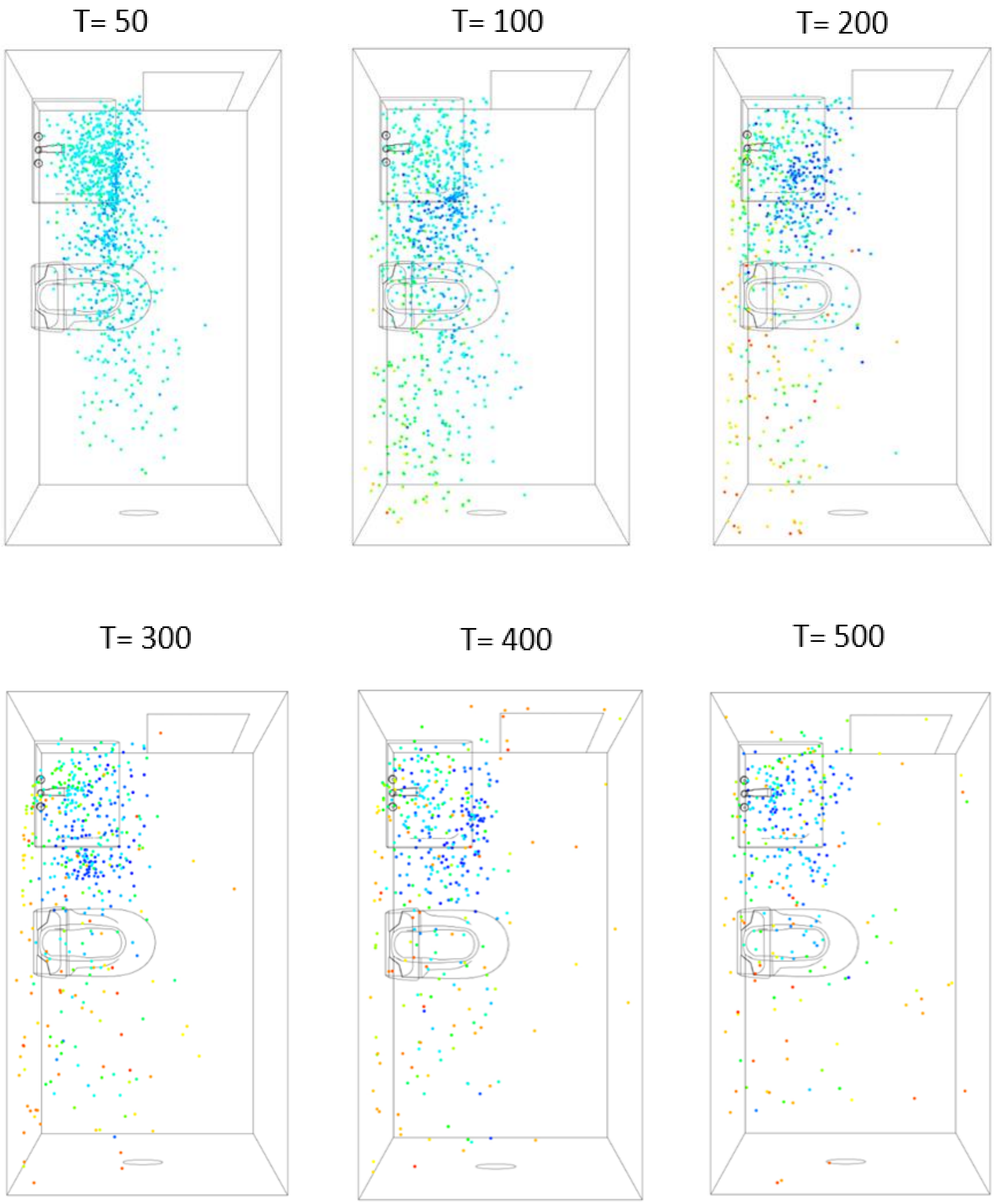


Figure 5. 7 Distribution of particle in the domain at different instants of time 50 s, 100s, 200 s,300 s, 400 s and 500 s from the time of injection at the washbasin. The particles are colored based on their height, such that red particles are close to the ceiling, green particles are at the height of a person and blue particles are close to the floor.

5.4.2 Injection at the center of washroom

After injecting in the secondary flow where we saw a lot more particles are harboring in the recirculation. Now we injected the particles in the primary flow which is center of the washroom. Figure 5.8 shows the number of particles at different instance of time. Here we have injected the particles at the center of washroom. So, the length to travel the particle will be almost half as compare to injection at center of washbasin. If the length of travel is reduced the primary flow time will be reduced by the half. The first particle will go out in 25 seconds.

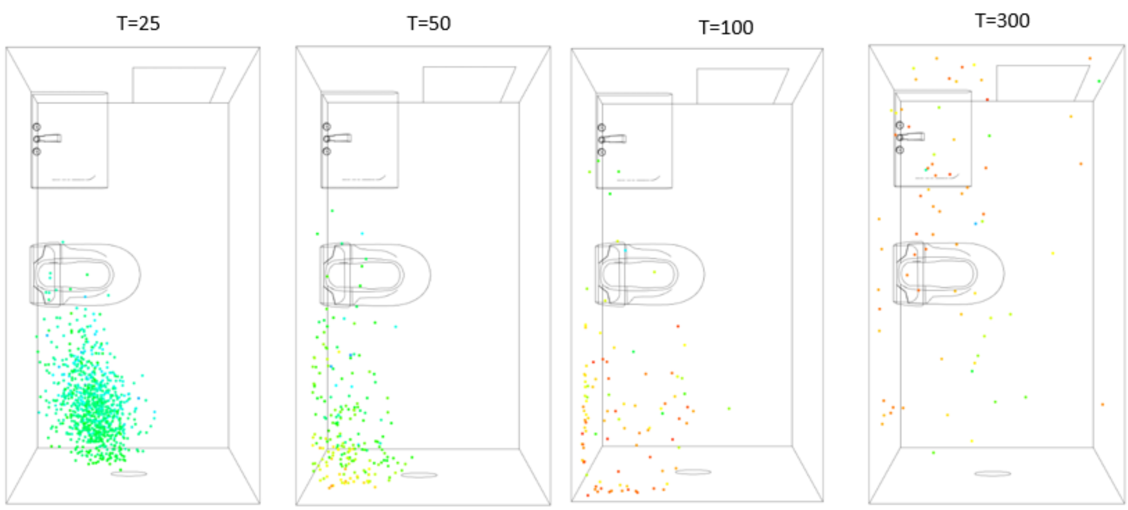


Figure 5. 8 Particles injected at the toilet seat are tracked over time in terms of their spatial distribution and height (shown by colour)

Since the injection is done in the primary flow. The most of the particles will soon go out the exhaust. We can see after 100 seconds very few particles left in the primary flow. After 300 seconds or later very less particles are left inside the washroom. But there are some particles which entered in the ceiling level. The entire ceiling is acting as reverse flow and particles will spend more time here.

5.5 Computation of particles

5.5.1 Computation of particles in the entire washroom

Till now we have seen only qualitative picture of the particles and position of particles in different time. We have also computed the how many particles are actually harbor in the secondary flow. We have written a UDF code to compute the exact number of particles in the different zones mentioned in Table 4.3. Figure 5.9 Shows the amount of mass in percentage after different time in both the injections. When the Particles are injected in the primary flow then after 25 seconds the particles started going out from exhaust. After 50 seconds only 20% particles are left inside the washroom. After 500 seconds or later only 1.5% of the mass is left in the domain.

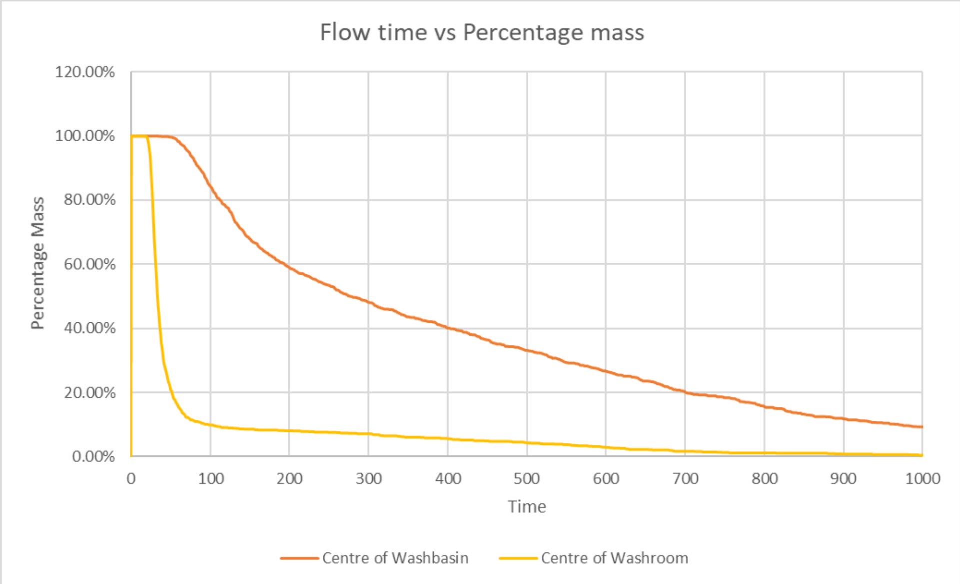


Figure 5. 9 Graph of percentage mass at different time in the entire domain orange graph shows when injection is done at center of washbasin and yellow shows when injection is done on center of washroom

Now the behavior of injection in the secondary flow which is represented by orange line in figure 5.9. If we see behavior of this injection, we can observe that after primary flow time which is 50 seconds 100% of the particles are present. After the 170 seconds still 60% particles are left. But the ideally as per the air change per hour calculation, in 170 seconds the entire air should be replaced by the fresh air. But that's not seems to be true. There are some particles which are trapped inside the dead zones.

5.5.2 Computation of the particles in different zones

We have seen above that huge mass is left after the ACH also. It becomes necessary to compute where are these particles which are tapped. As mentioned above we have computed the percentage of particles in different zones. Figure 5.10 shows the number of particles in the zone-1 and zone-2 respectively when the injection is done at the washbasin area.

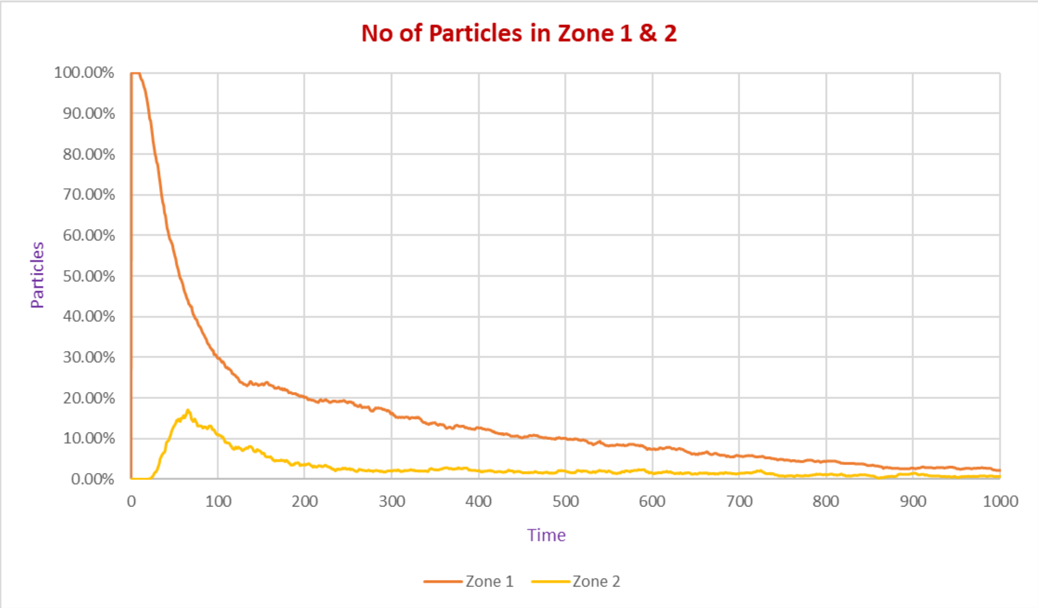


Figure 5. 10 Number of particles in zone-1 and zone-2 as percentage of total particles injected. Orange plot shows particles in zone 1 and yellow graph shows particles in zone 2

We can clearly see that the initially when we there will be injection at the center of the washbasin there will be around 100% particles in the zone-1 and zone 2 will be almost zero. As the time will pass the number of particles will come towards the zone-2 and they will get out of the exhaust. According to the flow field of the washroom, after some time zone-2 will be free from the particles and zone-1 will still harbor some particles.

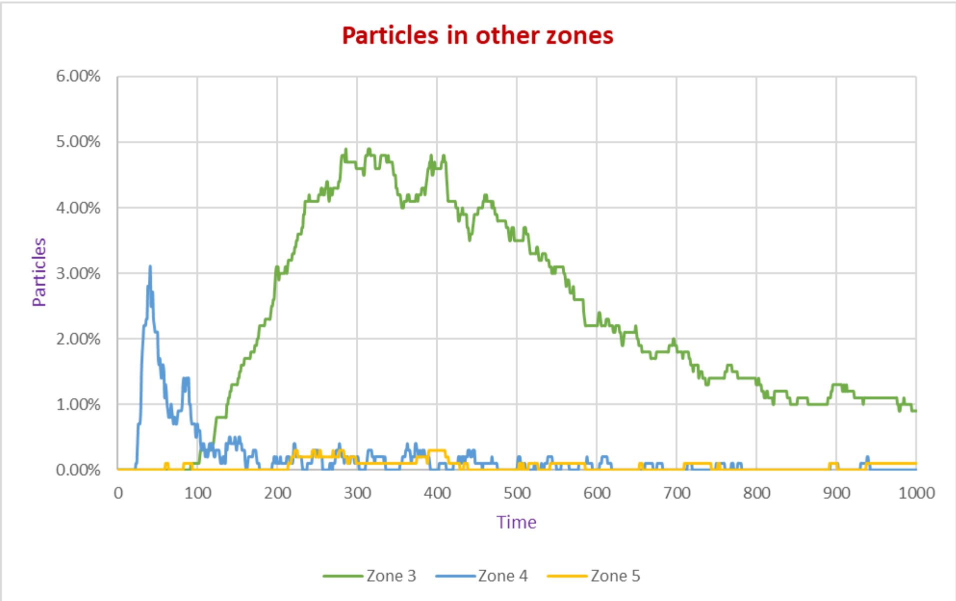


Figure 5. 11 Number of particles in zone-3,4,5 which are represented by green, blue and orange color graph respectively.

If we see the number of particles in other zones also, we can see that there are some particles which are trapped in the ceiling and other zones also. Figure 5.11 clearly shows are percentage of total particles which are left in zone-3, zone-4 and zone-5. The zone-3 which represents the ceiling has a greater number of particles.

Chapter 6

Secondary flow mitigation

From the above chapters it is clear that secondary flow or the dead zones are the big concerns for any enclosed space. There is need to remove those zones. For that purpose, we have tried to change the velocity of the exhaust in the 3D flow. If we have also tried to removing these recirculation zones with the help of door angles. We can disturb the original flow by deflecting flow towards the dead zones by making the door at angle. The effect of both the methods can be seen in this chapter.

6.1 Effect of the exhaust velocity

6.1.1 Two- dimensional case

We have kept the different exhaust velocity as 0.1 m/s, 0.2 m/s and 0.4 m/s apart from 0.3 m/s taken earlier. Figure 5.1.1 shows the flow pattern in the different cases.

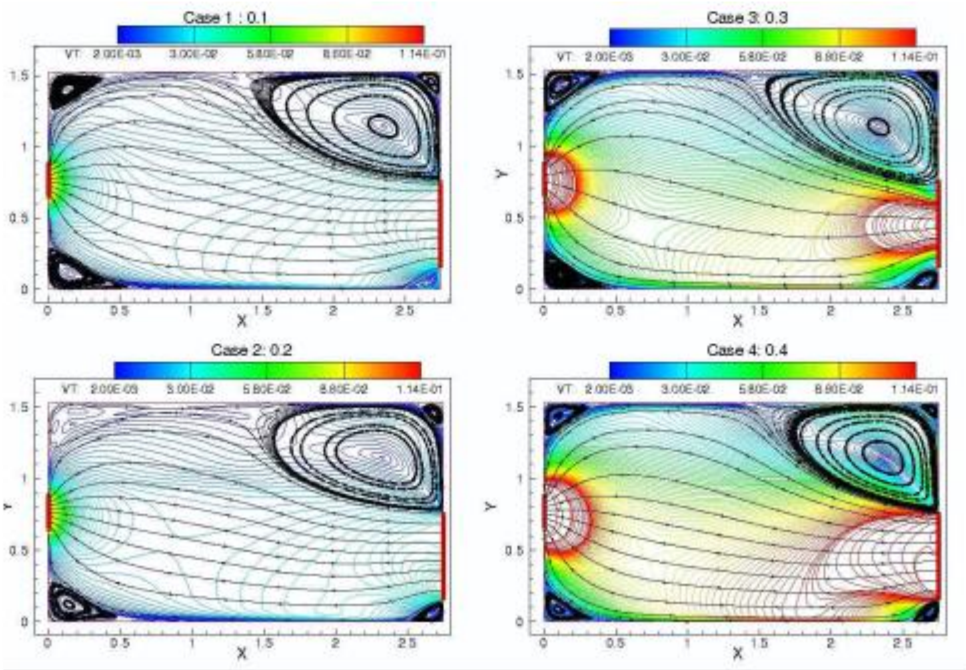


Figure 6. 1 Flow pattern in different exit velocities

From the above results, we found out that increasing the exit velocity may not eliminate the dead zones from the washroom. There is no major change in the flow pattern also. But when we have performed the different exhaust velocity in 3D case, we found some effect of the exhaust fan on the flow pattern. This is explained in next section.

6.1.3 Three- dimensional case

As in the case of varying the exhaust velocity, we have done some variation in the exhaust velocity in 3D case also. For the analysis 1 m/s, 1.5 m/s, 2 m/s, 3m/s velocity was kept at the exhaust. We have taken the x-velocity contours 1.8 m plane which includes the exhaust fan. In the figure 6.2 the while portion in contour shows the reverse direction of the flow. We did not find any major change in the recirculation zone which can be seen from the figure 6.2. Same thing was observed in the 2D case also.

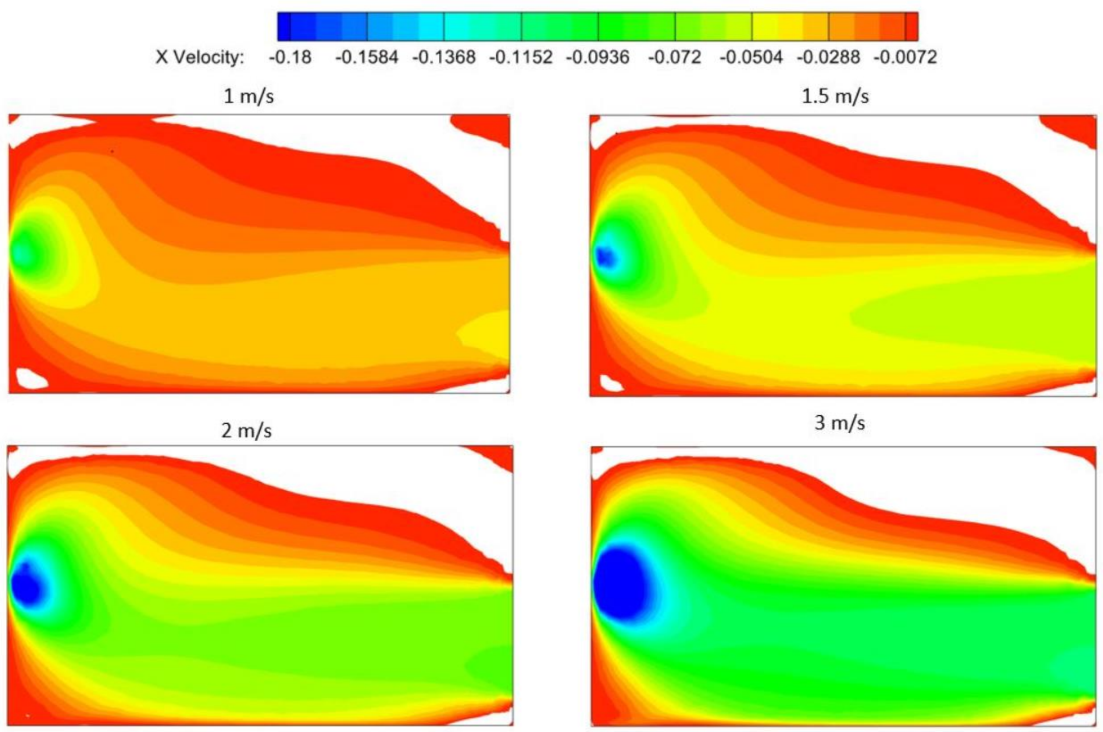


Figure 6. 2 Effect of exhaust fan velocity in 3D case

But when we have performed the particle analysis in the different exhaust fan velocities. We found some changes in the exit of the mass flow. The particles are exiting at the faster rate when the exhaust fan velocity is more. But in this case also the total mass flow rate will also become constant after 500 seconds or later. This shows that the fan with higher velocity may be able to remove the particles at faster rate but it will not be able to remove all the particles from this secondary flow zones.

Figure 6.3 shows that when exhaust velocity is 3 m/s then the mass will go faster from the exhaust as compare to velocity at 1.5 m/s. If we observe the decay of mass in both the cases. we have found that both the cases are following the exponential decay. The mass curve is following the function of flow time and the recirculation time.

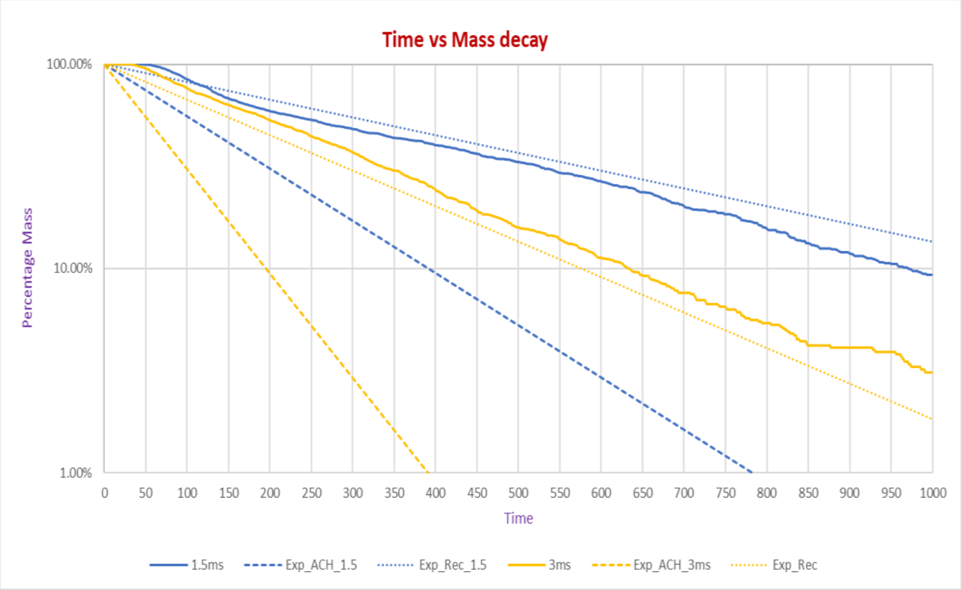


Figure 6. The rate of ventilation as indicated by the decrease in the particle mass remaining in the washroom, as a fraction of the total initial mass of particles injected at t = 0. The data for fan exit velocity of 1.5 m/s (blue) and 3.0 m/s (yellow).

If try to plot generalize the formulation for exponential decay of mass in the washroom. It comes out to be

$$\%M = e^{-(T/Rec)} \tag{6.1}$$

Where, $\%M$ = mass left at time T

T= time in second

$$ACH \ \alpha \ \frac{1}{Fan\ velocity} \tag{6.2}$$

Now if we try to plot this exponential decay of mass. We can clearly say that they are more or less following the same path as get from the simulations. The dashed blue line is representing the exponential decay for the fan velocity of 1.5 m/s and orange dashed line.

Secondary flow mitigation

From the above discussion we have seen that the higher fan velocity will still not be able to remove all the particles from the washroom. It becomes necessary to further study the effect of fan velocity on number of particles in primary and secondary flow zones. Figure 6.4 shows the effect of fan velocity in the zone-1. Here also both the graphs are following the exponential decay. The particles more or less follow the same as exponential decay.

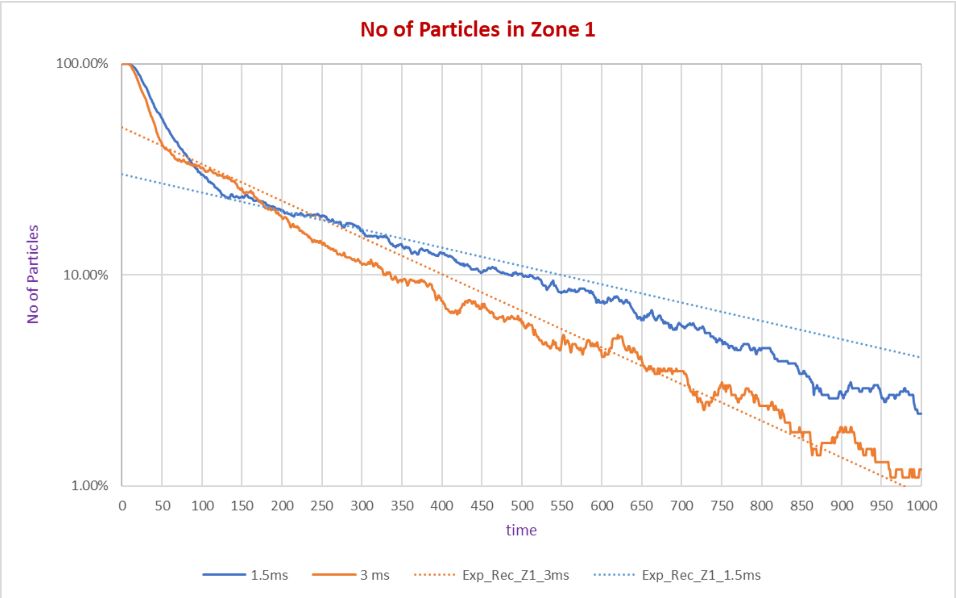


Figure 6. 3 Comparison of the particle numbers in semi-log plot in zone 1 for fan exit velocity of 1.5 m/s (blue) and 3.0 m/s (orange).

The exponential decay of the flow can be given by the formula

$$\%M = Ne^{-(T/Rec)}$$

Where N= number of particles injected at the start

If we observe both the curve then we can see that the fan velocity will definitely remove more particles from the zone-1 but still significant number of particles will be left in the secondary flow.

Now if study we same thing in the zone-2 which is in the primary flow. Here the effect of fan velocity will be more. The particles from the primary flow will go out faster. Both the curves are following same trend in removing the particles. Since the graph has lot more fluctuation so, it is be hard to predict the analytical solution for this.

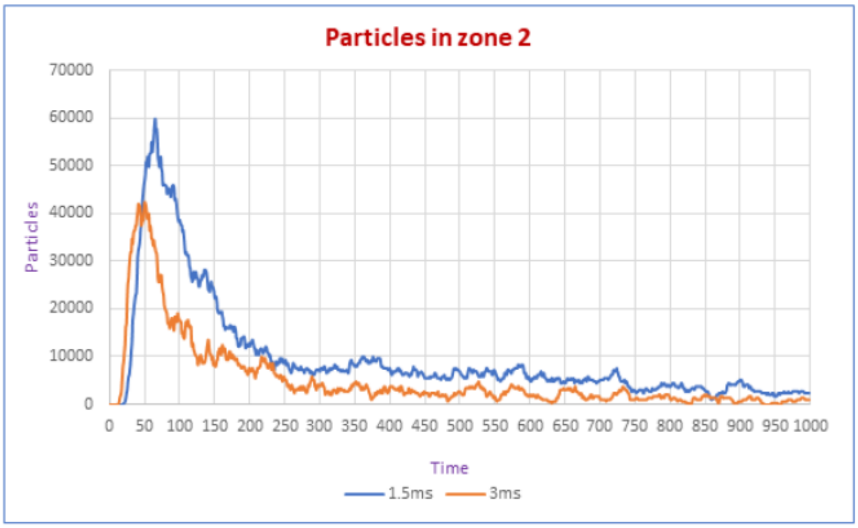
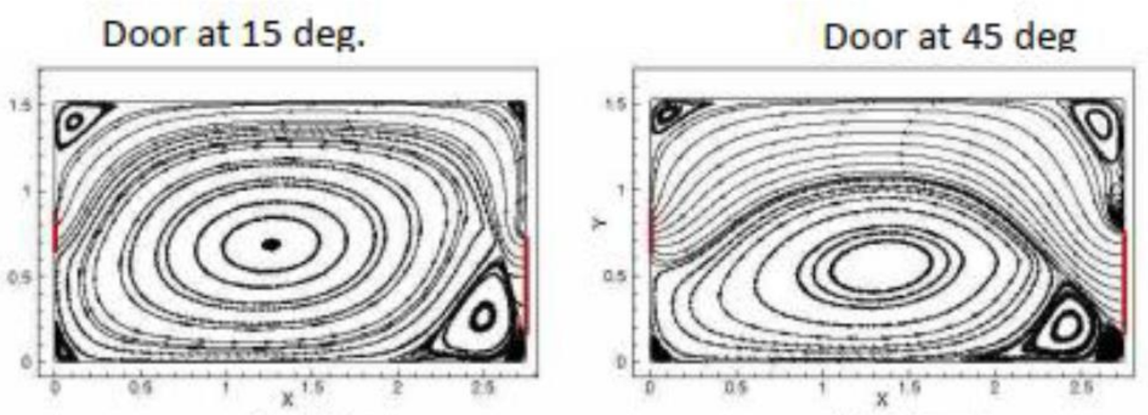


Figure 6.5 Comparison of the particle numbers in zone 2 for fan exit velocity of 1.5 m/s (blue) and 3.0 m/s (orange).

6.2 Effect of Door at angle

From the above discussion it is clear that the increase the velocity is not the complete solution to remove the particles from the secondary zones. Therefore, we need to adopt some other method to remove the particles from this zone. For the analysis we have 2-dimensional analysis for this. 3- dimensional analysis is not performed at door at angle due to the mesh constraints in ANSYS FLUENT Student version.

To mitigate the secondary from the dead zones, we have to take doors at an angle of 15 deg. and 45 deg. When the door is partially closed, it will divert the flow into the corner. It will eliminate the dead zones. But the partial door will create some secondary flow behind the door, which may not be the area of concern.



Secondary flow mitigation

Figure 6. 4 Flow pattern in a partial closed door (a) 15-degree door (b) 45-degree open door

From the above results, it is clear that CFD can give the optimum door angle for the better circulation of the air inside the washroom.

Chapter 8

Conclusion and Future work

This study introduces us to the importance of the flow visualization of the aerosol particles inside any closed vicinity. Understanding aerosol transport in different environments is of critical importance to COVID-19 mitigation measures. In the typical washroom of individual households, office complex, hospitals, malls, the dead zone near the washroom is the potential risk zone for the infection. The analysis is done using the CFD techniques and found out that the CFD is a great way to predict the flow. There are various conclusions can be drawn from this study. These conclusions are given below:

1. Inside the enclosed space there will be formation of the secondary flow. This will depend upon the Inlet and outlet boundary conditions.

2. The residence time of these secondary flow will be more than the primary flow. The secondary flow residence time can be 10-15 times more than the primary flow time for present configuration.

3. The roof /ceiling will act as recirculation zone, where the particles will stay long. If any ceiling fan is present then it can be of a greater concern.

4. The secondary zones will harbor a greater number of particles.

5. The presence of fittings and furniture inside an enclosed space will create some extra secondary flow zones.

6. The smaller size of the particles will stay for more amount of time than larger size of the particles.

7. The number of particles left inside the room will also depend upon the location of the injection of particles.

8. The higher CFM of fan may not eliminate the secondary flow from the enclosed area. But it has impact in exchanging the inside air faster.

9. The secondary flow can be removed by changing the door angle. But there will be another secondary flow will form behind the door.

5.1 Future work

The same studies can be used in many places, like office space, meeting rooms, auditoriums. The above studies can be verified by the time-averaged images from the smoke videos using Particle Image Velocimetry (PIV) technique. Present study does not include the evaporation model. The future study can include evaporation model. The study can be further extended to include the different diameters of the particles at different locations. This study does not include the any person inside the enclosed space. The presence of person inside the space will largely disturb the flow. In future the mannequin can be included. Further studies in this area can include the effect of temperature and humidity conditions.

Appendices

In this chapter, we will explore some more two-dimensional simulations performed in the real laboratory setup and the office setup inside the IIT Bombay campus.

HCFD Lab Simulations

The Hypersonic CFD lab of the IIT Bombay has two chambers which are interconnected. Initially we have performed the simulation considering only one room which has multiple inlet and one exhaust. The Figure 9.1 shows the setup of the lab considering only single room. In this lab there are multiple inlets which can be adjusted. There is one exhaust is also provided at the corner represented by outward arrow. The inlets are of different size. The inlet 1 & 2 is 1.06 m. Inlet 3 is 0.67 m and exhaust are 0.762 m. These simulations were also performed in the same methodology mentioned in the chapter 4. The boundary condition at exhaust is given as 0.3 m/s which is calculated from the ACH of the fan and dimensions of the room.

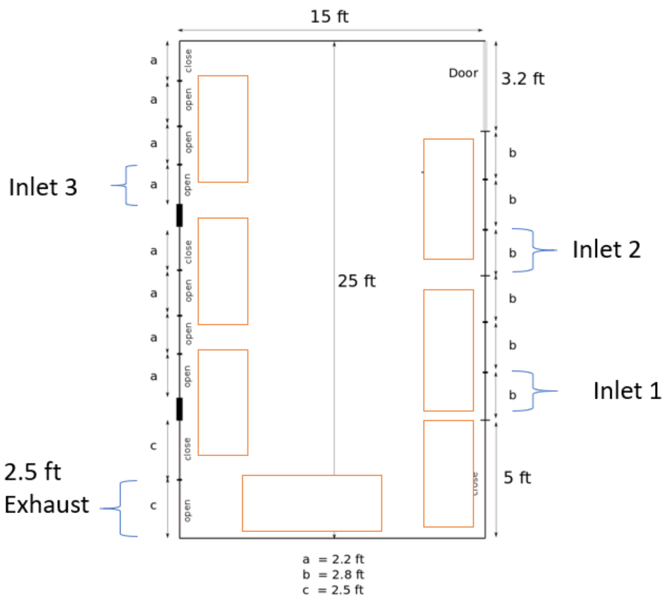
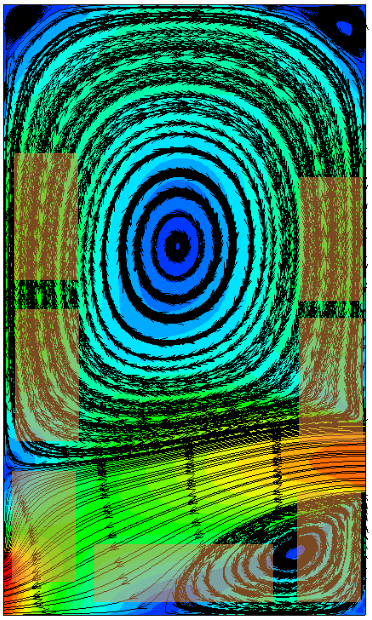


Figure 9. 1Lab layout for single room in which multiple windows are considered as inlet and 2.5 ft window in bottom left corner is considered as exhaust.

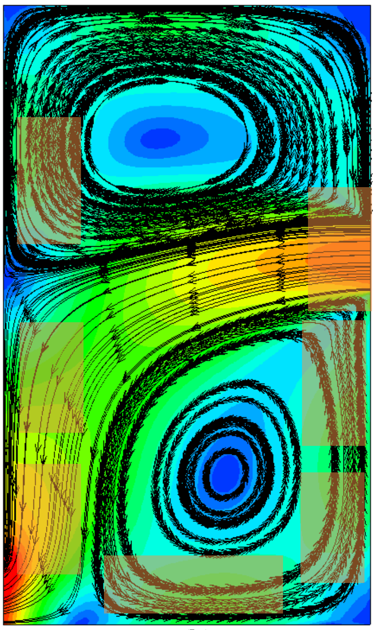
We have performed simulation by opening different inlets given in figure 9.1. In the first case only inlet 1 is opened and other are closed. The flow results are represented by Figure 9.2. In this condition almost entire room is becoming the recirculation zone.



(a)

Figure 9. 2 Shows the velocity contours and the streamlines plotted over the that contour to show the flow pattern for Case 1

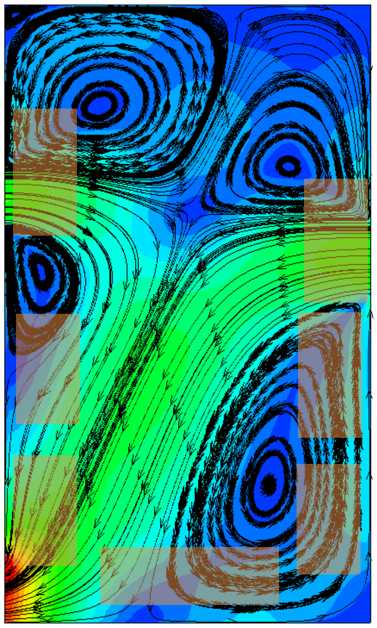
In the second condition we have opened the inlet 2 and inlet 1 is closed. The figure 9.3 shows the flow pattern for this case.



(b)

Figure 9. 3 Shows the velocity contours and the streamlines plotted over the that contour to show the flow pattern for Case 2

In the third case we have opened inlet 2, and 3 for the better ventilation. Figure 9.4 shows the flow patter for the third case.



(c)

Figure 9. 4 Shows the velocity contours and the streamlines plotted over the that contour to show the flow pattern for (a) case 1 (b) case 2 and (c) case 3

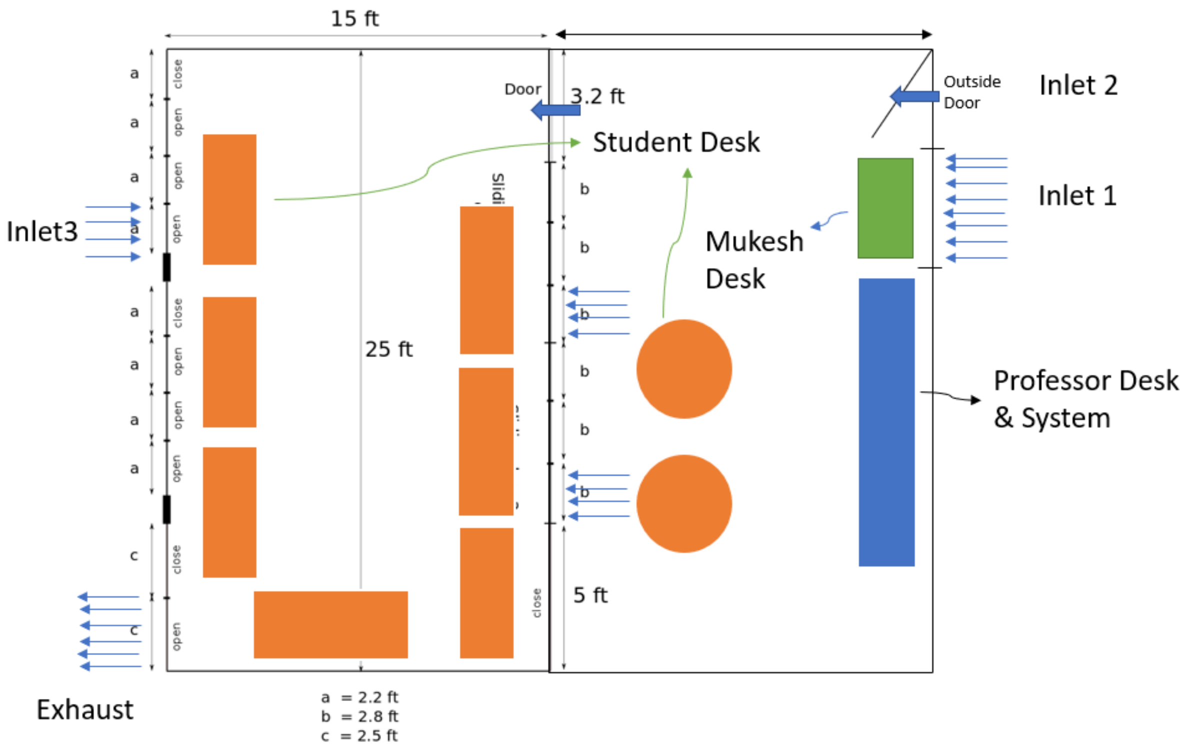
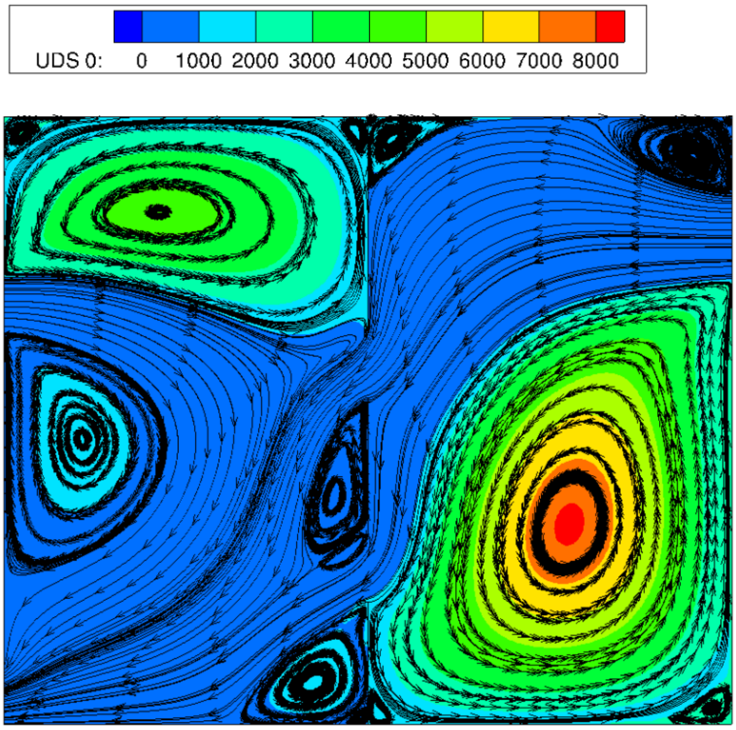
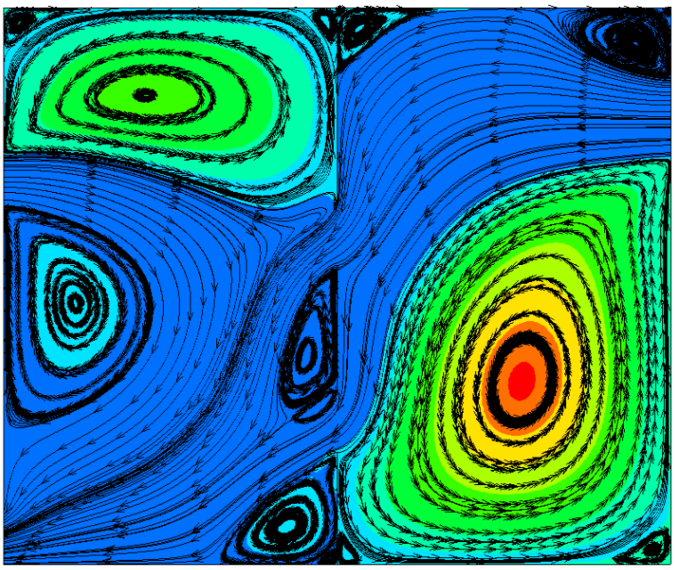


Figure 9. 5 Layout of HCFD lab considering both rooms

Now the we have performed the simulation considering both the rooms. Figure 9.5 shows the layout for both the rooms. Now for the full room conditions, we have changed the inlets but the location of exhaust is same. Figure 9.5 shows the different inlets and exhaust location. The simulations were performed by changing the location of the inlets. There are 3 types of combinations were performed. Figure 9.6 (a) shows that inlet 1 & 3 are open and rest of them are closed. We can see a big recirculation zone in the room 1. To eliminate this zone, we applied some changed in the inlet of the flow. Figure 9.6 (b) shows that the inlet 2 is partially open in order to eliminate the big recirculation zone of outer room. Now the inlet 1 is closed. We can see in the figure 9.6 (b) that the earlier zone was somewhat reduced. But this configuration created another recirculation zone near the inner door. To eliminate this zone, we have performed third combination of the boundary condition. In this we have opened inlet 2 & 3 and also opened inside door. We can see in the figure 9.6 (c) that there is still formation of some big recirculation zones but the residence time of these recirculation zone is reduced.



(a)

(b) 

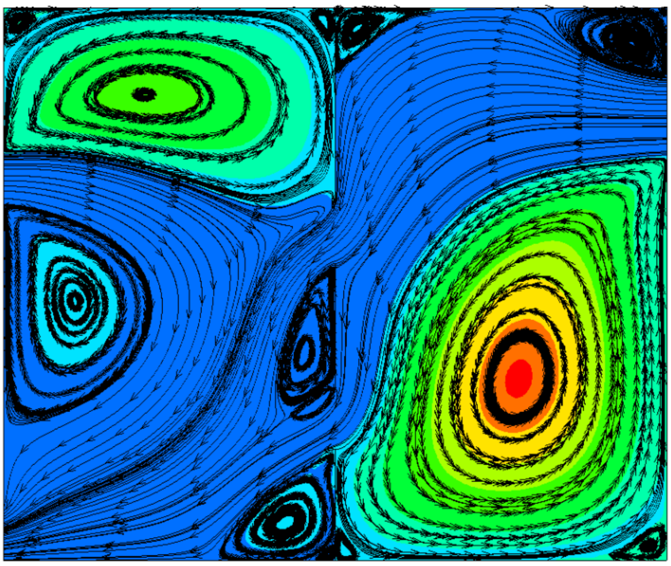
(c) 

Figure 9. 6 Residence time contour with velocity streamlines in which (a) shows residence contour for case 1 (b) contour for case 2 and (C) shows the residence time contour for case 3

The two-dimensional simulations are also performed inside the IRCC office campus of the IIT Bombay. The details of these simulations are given below.

IRCC Office simulations

The actual office is big to do simulation with student version of IRCC office. We have taken a section (Account section) of the office to find the flow pattern and secondary flow zones. In Actual there are multiple tables, chairs and cubicles are present. These things will highly disturb the flow. But the for the simplicity we have taken the wall boundaries only and neglected the chairs, tables and cubicles. Figure 9.3 shows the actual account section layout. Figure 9.4 shows the computational domain which is considered for the simulation.

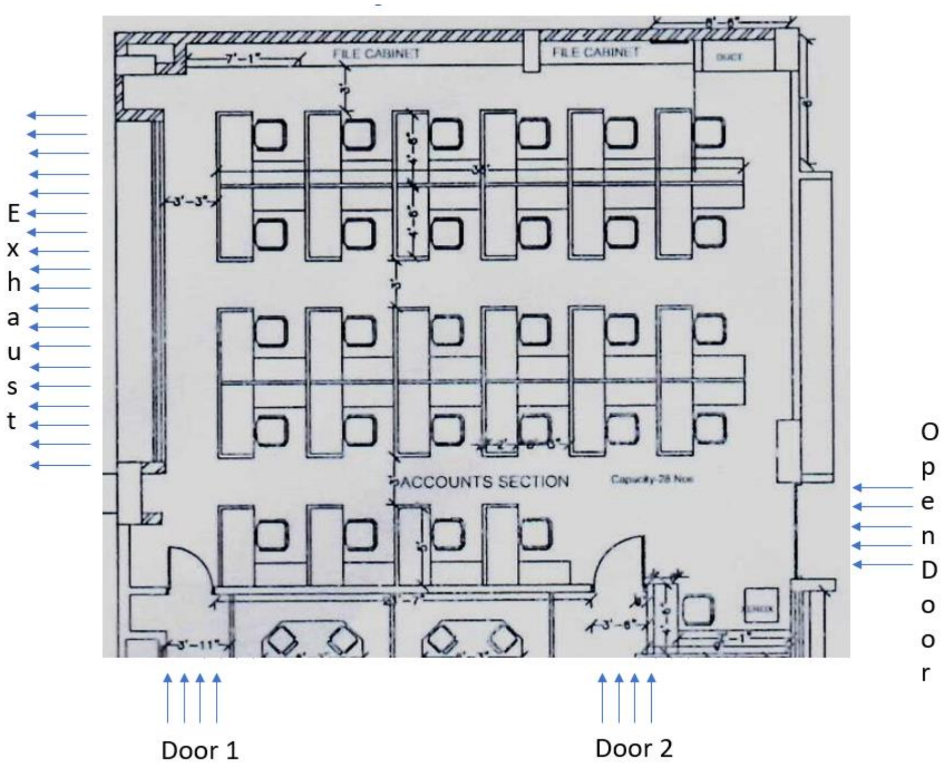


Figure 9. 7 IRCC office layout of account section

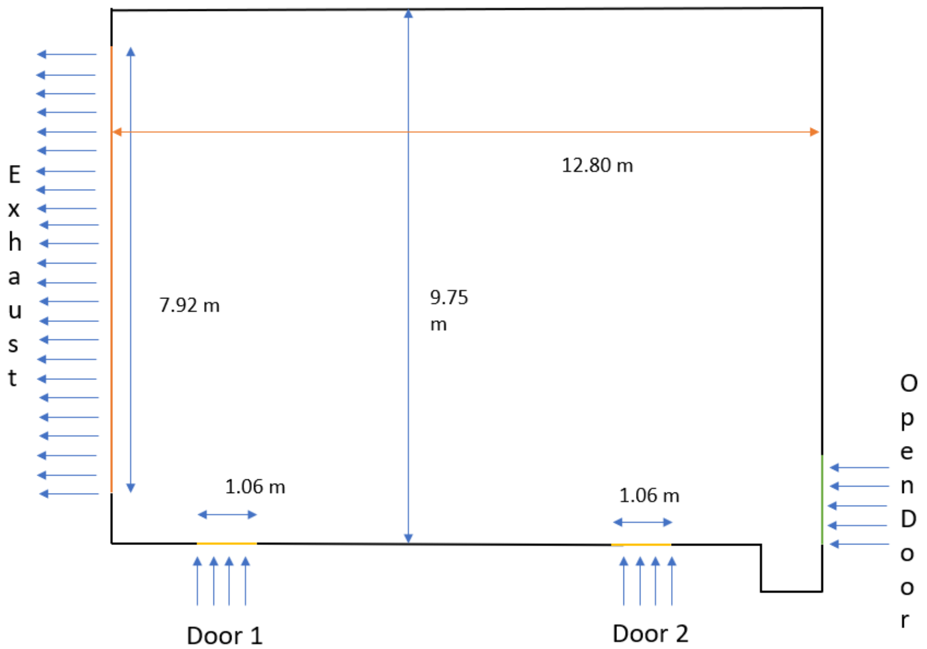


Figure 9. 8 Computational domain for IRCC office

The dimensions of the inlet and outlet are mentioned in the figure 9.4. Here the exhaust velocity is given as 0.1 m/s. The inlet is at zero-gauge pressure. The rest of the simulation conditions are same as mentioned in the chapter 4. The results of the simulations are as follows.

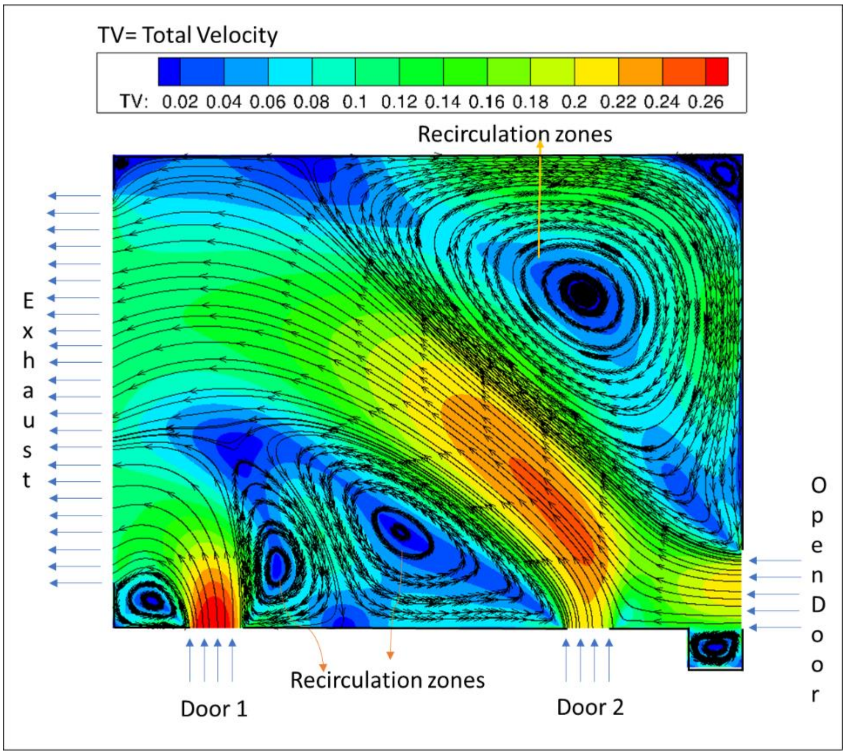


Figure 9. 9 Total velocity contours with velocity streamlines in office section

The residence time of this domain is represented by the figure 9.6.

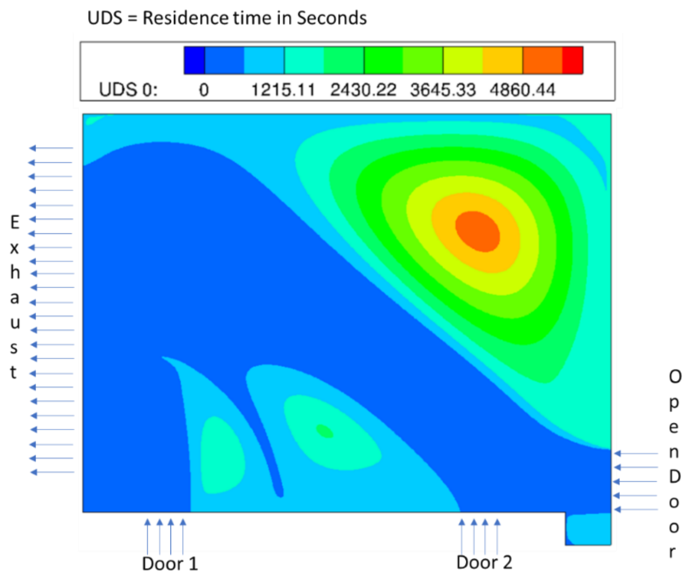
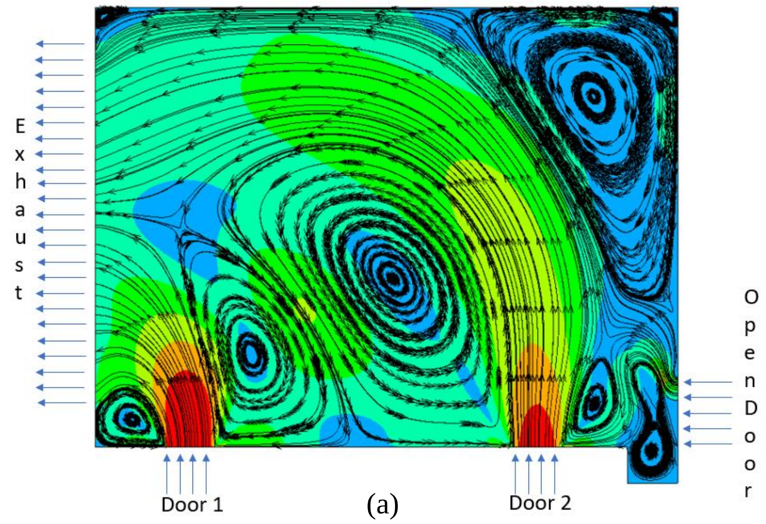
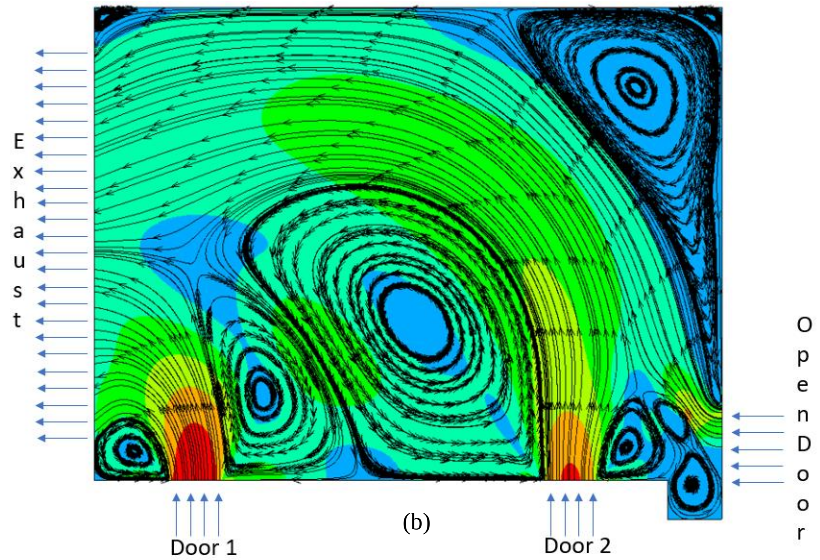


Figure 9. 10 Residence time contour of office section

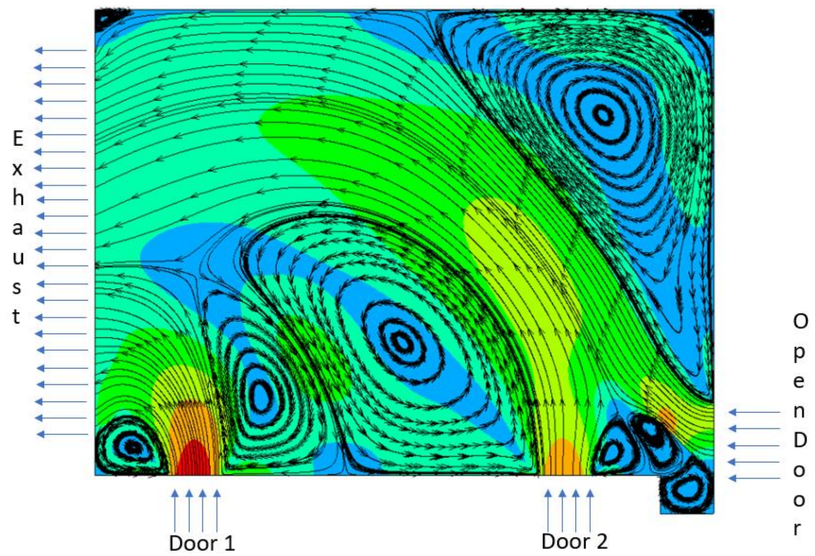
Since there is formation of the recirculation zone. We have also tried to reduce this recirculation with the help of changing the angle of the inlet. Flow is directed towards the recirculation zone which eventually reduces the size the recirculation zone. Figure 9.7 (a), (b), (c) represents the different door angle 45 ,30, 15 degrees respectively. The angle of door can optimize with minimum size of the recirculation zone.



(a)



(b)



(C)

Figure 9. 11 Total velocity contours with velocity streamlines in which (a) represents the contours when door at 15- degree angle (b) represents the contours when door at 30- degree angle (c) represents the contours when door at 45- degree angle

Bibliography

Abuhgazy, M., 2020. Numerical investigation of aerosol transport in a classroom with relevence to COVID-19. *Physics of Fluid 32.*

Ahmadi, A. L. a. G., 1992. Dispersion and Deposition of Spherical particles from point source in a Turbulent Channel flow. *Aerosol Science and Technology,* pp. 209-226.

Ai Min Yang, Y. F. L., 2017. Numerical study on aerodyanmics characteristics of high speed trains with considering thermal flow coupling effects. *Journal on Vibroengineering,* pp. 5606-5626.

Aliahmadipour, M., 2017. Airflow Simulation of HVAC system in a compartment of the passanger. *Applied thermal Engineering,* pp. 973-990.

Anon., 2020 . *Fluent Theory Guide R.2.,* Canonsburg P.A: ANSYS Inc.

George, C., 2016. Numerical simulation of HVAC system functionally in a socio economic cultural building. *Procedia technology,* pp. 535-542.

Haitao Want, M. L., 2014. Performance evaluation of air distribution systems in three different China railway high-speed trains cabin using numerical simulations. *Buidling Simulations,* pp. 629-638.

Li, L. Z. Y., 2012. Dispersion of Cougehed particles in a fully-occupied high-speed rain cabin. *Buidling and Environment,* pp. 58-66.

Peyvan, J. K. A., 2021. Computer simulation of SARS CoV-2 contamination risk in large dental clinic. *Physics of Fluids,* p. 033328.