

STUDYING THE VENTILATION PATTERNS IN A COVID-19 MULTI-PATIENT WARD USING CFD

*Report submitted to the SASTRA Deemed to be University
as the requirement for the course*

MEC300 MINI PROJECT WORK

Submitted by

Naimish Mani B
(Reg. No.: 122009154)
Shyam Ganesh R
(Reg. No.: 122009228)

December 2021



SCHOOL OF MECHANICAL ENGINEERING

THANJAVUR, TAMIL NADU, INDIA – 613 401



SCHOOL OF MECHANICAL ENGINEERING
THANJAVUR – 613 401

Bonafide Certificate

This is to certify that the report titled “**Studying The Ventilation Patterns In A Covid-19 Multi-Patient Ward Using CFD**” submitted as a requirement for the course, **MEC 300 Mini-project Work** for B.Tech. Mechanical programme, is a bonafide record of the work done by **Mr. Naimish Mani B (Reg. No.122009154)** and **Shyam Ganesh R (Reg. No.: 122009228)** during the academic year 2021-22, in the School of Mechanical Engineering, under my supervision.

Signature of Project Supervisor :

Name with Affiliation : S. Ramaswami, SoME

Date :

Mini-project Viva-você held on 13th January, 2022

Examiner 1

Examiner 2



**SCHOOL OF MECHANICAL ENGINEERING
THANJAVUR – 613 401**

Declaration

We declare that the report titled “**Studying The Ventilation Patterns In A Covid-19 Multi-Patient Ward Using CFD**” submitted by us is an original work done by us under the guidance of **Dr. S. Ramaswami, SAP, School of Mechanical Engineering, SASTRA Deemed to be University** during the seventh semester of the academic year 2021-22, in the **School of Mechanical Engineering**. The work is original and wherever we have used materials from other sources, we have given due credit and cited them in the text of the report. This report has not formed the basis for the award of any degree, diploma, associate-ship, fellowship or other similar title to any candidate of any University.

Signature of the candidate(s) :

Name of the candidate(s) : Naimish Mani B Shyam Ganesh R
Date : 22nd December, 2021

Acknowledgments

At the prime beginning of this report, we would like to extend our sincere and heartfelt obligation towards all the persons who have helped us in this tentative situation . Without their active guidance, help, cooperation and encouragement, we would not be able to complete this project.

It is our pleasant duty to express a deep sense of gratitude and respect to our Honourable Vice Chancellor Dr. S.Vaidhyasubramaniam, SASTRA Deemed to be University the paramount of the institution, for his patronage and Dr. R. Chandaramouli, The Registrar, SASTRA Deemed to be University for providing us the necessary facilities to carry out this research work remotely.

We express our sincere thanks and gratitude to Dr.S.Pugazhenthii, Dean, School of Mechanical Engineering, SASTRA Deemed to be University, for his support in the accomplishment of our project work. Furthermore, we would like to express our sincere gratitude to our project guide Dr. S. Ramaswami for his continuous support of our project work, his patience, motivation, enthusiasm, and immense knowledge. We are extremely indebted to our project mentor for his conscientious guidance and encouragement to accomplish this project.

We extend our gratitude to SASTRA Deemed to be University for giving us this opportunity in developing the project and to the people who have willingly helped us out with their abilities.

Any omission in this brief acknowledgement does not mean lack of gratitude.

Table of Contents

Title	Page No.
Bona-fide Certificate	ii
Declaration	iii
Acknowledgements	iv
List of Figures	v
List of Tables	vi
Abstract	ix
1. Introduction	
1.1. General Overview	1
1.2. Motivation	1
1.3. Iterative Design Process	1
1.4. Software Used	2
2. Literature Survey	3
3. Objectives	4
4. Methodology	
4.1. Duct Design	5
4.2. CAD Model	6
4.3. Mesh Generation	6
4.4. Mathematical Modelling	7
4.5. OpenFOAM case setup	10
4.6. Post Processing	11
5. Results and Discussion	
5.1. 1_inlet_1_outlet	13
5.2. 2_branch_pair_shifted	14
5.3. 3_wall_4_ceiling	16
5.4. 3_wall_5_ceiling	17
6. Conclusions and Future Work	20
7. References	21

List of Figures

Figure No.	Title	Page No.
4.1	CAD model of a hospital room	6
4.2	Mesh	7
5.1	1_inlet_1_outlet	14
5.2	branch_pair_shifted	15
5.3	3_wall_4_ceiling	17
5.4	3_wall_5_ceiling	18
5.5	Comparison	19

List of Tables

Table No.	Table name	Page No.
4.1	Dimensions of the hospital room and bed	6
4.2	Finite volume discretization schemes	9
4.3	Initial and Boundary conditions	10
4.4	Schemes used in OpenFOAM	11

ABBREVIATIONS

CFD	Computational Fluid Dynamics
NCDC	National Centre for Disease control
FVM	Finite Volume Method
RANS	Reynolds Averaged Navier Stokes
CAD	Computer Aided Design
HEPA	High Efficiency Particulate Air
UV	Ultra Violet
STEP	Standard for The Exchange of Product model data
ACPH	Air Changes Per Hour

NOTATIONS

Q	Volumetric flow rate
A	Area
T	Temperature
ν	Kinematic viscosity
ρ	Density
u	velocity
τ	Stress vector
p	Pressure
V	volume

ABSTRACT

Isolation wards are used in hospitals to isolate patients who have contracted contagious diseases like COVID-19, cholera and Ebola. Hospitals are required to implement containment strategies to prevent their staff and visitors from spreading the disease. One way in which respiratory disease outbreaks can be controlled inside a hospital is through the study of the natural ventilation patterns inside a hospital ward. Usually, during a disease outbreak, the National Centre for Disease Control (NCDC) publishes guidelines addressing the setting up of isolation wards in hospitals.

In this project, we design an isolation ward using FreeCAD, a 3D modelling software, in accordance with the NCDC guidelines for COVID 19. We subsequently use OpenFOAM, a finite-volume CFD solver, to study the ventilation patterns in the room. This project is carried out with free and open source software for greater transparency and reproducibility. We also make use of Gmsh for meshing, ParaView for post-processing and MPICH for parallelising our simulation.

CHAPTER 1

INTRODUCTION

1.1 General Overview

Over the last two years, right since the emergence of the COVID-19 pandemic, there has been increased consciousness of the importance of the airflow inside a hospital room, specifically when designing air conditioning and mechanical ventilation systems. Improper ventilation can result in the contamination of air, leading to the spread of infectious diseases from patients to doctors, nurses and other healthcare workers. Accurate prediction of air flow parameters and contaminant distribution can help in providing an effective ventilation system. Due to the prohibitive cost of laboratory experimentation, Computational Fluid Dynamics (CFD) has nowadays been increasingly employed to simulate the indoor climate. Among all the existing CFD tools, OpenFOAM is a popular open-source CFD simulator, which is an object-oriented C++ tool box specifically developed for the Finite Volume (FV) method. Through this project, we designed a COVID-19 isolation ward in accordance with government guidelines and studied the airflow in those wards. We subsequently made improvements to our ventilation system by adhering to the principles of iterative design.

1.2 Motivation

Historically, hospitals used to accommodate 3 to 4 patients per ward to provide treatment for infectious diseases. The ventilation mechanisms were designed to accommodate such a patient load in the hospital. With the onset of the COVID-19 pandemic and growing number of infections, new central government regulations mandate each district to have wards that can accommodate a minimum of 10 patients, and also specify certain operating conditions to be maintained inside the hospital premises. For the changed operating conditions, a customised ventilation mechanism is necessary, but very limited research has gone into designing such a system with the COVID-19 pandemic in mind. Generally, off-the-shelf licensed products which employ CFD principles to handle ventilation are being used by various stakeholders. But open-source CFD saves users a considerable license cost and provides users with full transparency of implementation and maximum freedom of customization. Therefore, we carried out this project by making use of only free and open source software packages.

1.3 Iterative Design Process

In general, the iterative design process is a cyclic prototyping procedure through which one tests, analyses and refines any given design or process. We apply this methodology to our problem by iterating through positioning of the ventilation ducts at various locations across the walls and ceilings of the COVID-19 isolation ward. During each iteration, we run the CFD simulation and

analyse the flow field in the ward. Using our inferences from the simulation, we further refine our positioning.

1.4 Tools and Software used

This project was wholly carried out using the following free and open source software packages:

- FreeCAD (v0.18), for designing the isolation ward
- Gmsh (v4.8.4), for meshing the CAD model
- OpenFOAM (v8 and v2012), for running the CFD simulation
- ParaView (v5.6.0), for post processing
- MPICH (v3.3.2), for parallelizing the simulation run

CHAPTER 2

LITERATURE REVIEW

In general, two main softwares are used for CFD simulation, ANSYS Fluent and OpenFOAM. Méndez, C. et. al. [6] have compared the results produced by OpenFOAM and Fluent is for flow through a cylindrical cavity. From their study, we can conclude that through use of a finer mesh for OpenFOAM, we can get results similar to that from Fluent, without having to break the bank. Wang, C. et. al. [1] have employed OpenFOAM's RANS solver to study the airflow in a room at low velocities (2.7 m/s). From their study, we can conclude that we can get by without modelling turbulent flow and still get a close approximation of the actual flow. Córdova-Suárez, M. et. al. [5] and Beggs, C. et. al. [7] design rooms in accordance with standard guidelines for their respective applications, and use CFD to verify these designs. From these studies, we can conclude that placing multiple vents in different parts of the room can be a very effective solution at getting rid of biological and chemical contaminants from the air. Welahettige, P. et. al. [4], Shih, Y. et. al. [8] and K.W.D. Cheong et. al. [9] predominantly concern themselves with various measures to control the flow of air inside hospital wards, while also looking at different methods to deal with pathogen particles. From [8], we can conclude that the disturbances in airflow due to the movement of healthcare specialists in our room can be neglected. Similarly, Welahettige, P. et. al. [4] look at the effects of partitioning sections on the ventilation systems, and how they are effective in trapping pathogens into a chokepoint. Through K.W.D. Cheong et. al. [9] CFD analysis, we can also see the effects of the position of furniture on the ventilation of a room, and so can plan our CAD model more effectively. They also highlight that positioning air inlets near the health-care workers' station and the outlet near the most infectious patients can significantly increase the medical efficiency of the ventilation system. Juan Ren et. al. [10], through their CFD analysis, found that most ventilation systems designed according to the WHO guidelines are only effective at transporting lightweight particles (diameter < 20 microns), and that it is important to regularly manually clean and disinfect regions where there is low air velocity to prevent contaminants from aggregating in those regions. Vladimir P. Zhdanov et. al. [12] and Luis A. Anchordoqui et. al. [13] discuss the governing equations that can be used to model the virion particles that carry and transmit viri such as COVID-19.

CHAPTER 3

OBJECTIVE

This project aimed to achieve two objectives:

1. Designing an effective ventilation mechanism for a hospital ward in accordance with NCDC guidelines.
2. Developing a procedure pipeline to achieve (1) using only open source software packages.

The NCDC guidelines mandate a minimum of a 10-bed isolation ward established at the district level, with at least a 1 meter (3 feet) separation between the beds. They also mention that a minimum space of 2000 sq. ft. area should be used as required. The air flow rate is also required to be at least 12 air changes per hour with a filtration system present at the output. If an air conditioning system is present, it is required to be a stand-alone system not integrated with the overall centralised cooling system.

We subsequently design all wards to meet these minimum requirements. For the sake of simplicity, we assume the room dimensions to be 60 ft x 40 ft x 10 ft. To improve editability in the iterative design process, we save our geometries as STEP files.

Although this project does not concern itself with the filtration of air once it exits the outlet vents, passing the output through HEPA and UV filters have been proposed in literature and this problem is actively being studied by experts.

An important factor contributing to the effectiveness of a given ventilation system is the ability of said system to redirect flow from the nurses' station to the rest of the room. This is done to reduce the probability of the nurses stationed in the ward from contracting the virus. Apart from this, we also note the absence of large domains without flow in favorable light when objectively quantifying the effectiveness of a given ventilation system.

CHAPTER 4

METHODOLOGY

To study the ventilation patterns in a COVID-19 multi-patient ward using CFD, we first need to define the floor plan of the multi-patient ward, including 10 patient beds with tables and a nurse station. Next, we need to determine the dimensions of the ventilation ducts. With this, we can proceed to define the governing equations of flow in our system.

4.1 Duct Design

According to the NCDC guidelines, the room is required to undergo a minimum of 12 air changes per hour (ACPH). Given a total room volume of 24000 ft³, we calculate the minimum and maximum possible area for a ventilation duct to ensure the required ACPH value. The calculations are as follows:

$$ACPH = \frac{60Q}{Volume}$$

$$\Rightarrow 12 = \frac{60Q}{24000}$$

$$\Rightarrow Q = 4800CFM$$

$$Area = \frac{Q}{v}$$

$$A_{max} = \frac{4800}{1000} = 4.8 \text{ sq ft}$$

$$A_{min} = \frac{4800}{1500} = 3.2 \text{ sq ft}$$

From this, we can see that the range of possible areas for a vent is between 3.2 and 4.8 sq. ft. We select the area of the vent to be 4.8 sq ft, and allot its dimensions as 3 ft x 1.6 ft.

4.2 CAD Model

After calculating the duct dimensions, we designed a CAD model of the room to perform the ventilation analysis. The room features a nurse station, 10 patient beds (separated by a distance of at least 3 feet) accompanied by bedside tables. The CAD model was prepared in FreeCAD in accordance with the design specifications found in Table 4.1. A screenshot of the base validation case is given in Figure 4.1.

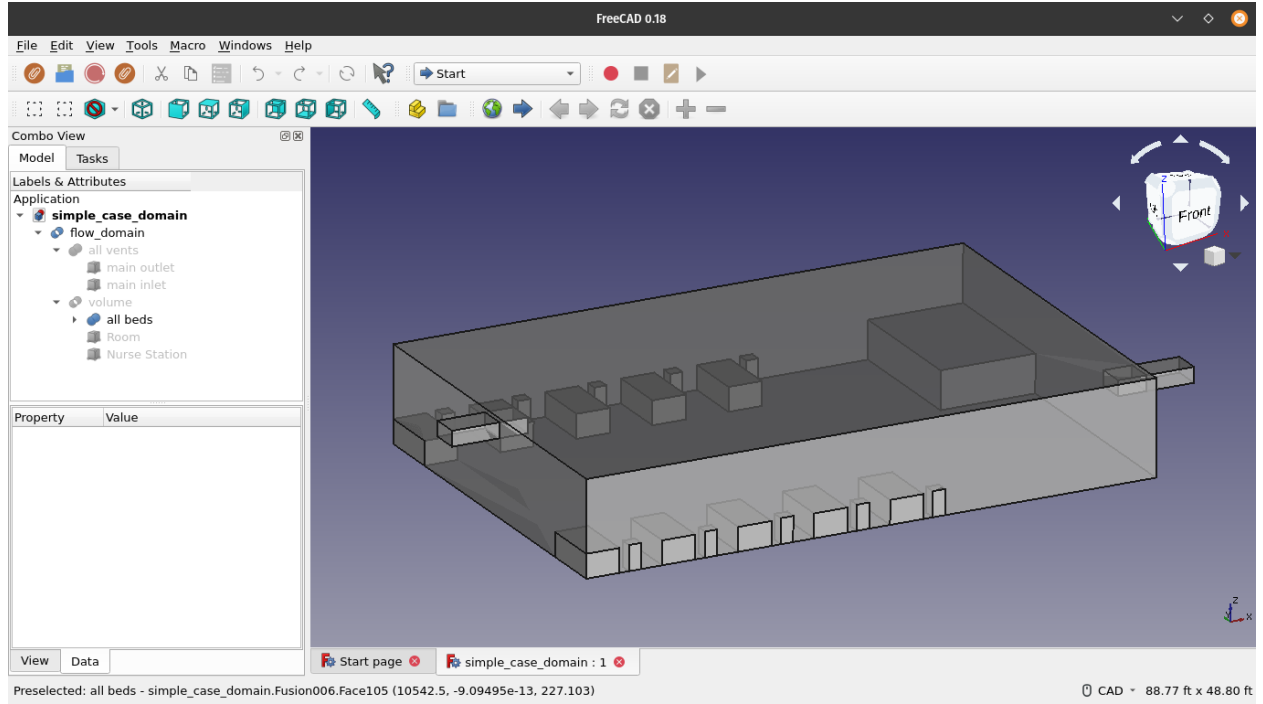


Fig 4.1 CAD model of the hospital room

Name	X direction length (ft)	Y direction length (ft)	Z direction length (ft)
Room	60	40	10
Patient Bed	6.5	3.5	2.5
Patient bedside table	1.3	1.3	2.62
Nurse station	15	10	4
Ventilation duct	3	8	1.6

Table 4.1 Dimensions of the hospital room and bed

4.3 Mesh Generation :

To perform any form of computational analysis, we are required to discretize our field into infinitesimal units. This is achieved through meshing. We make use of Gmsh, an open source meshing tool, to generate the mesh from the CAD model. The mesh is refined till it has at least 250,000 cells. The validation case makes use of simple tetrahedral cells, whereas later cases make use of hexahedral cells, which help achieve higher accuracy. A screenshot of the mesh for the validation case is given in Figure 4.2.

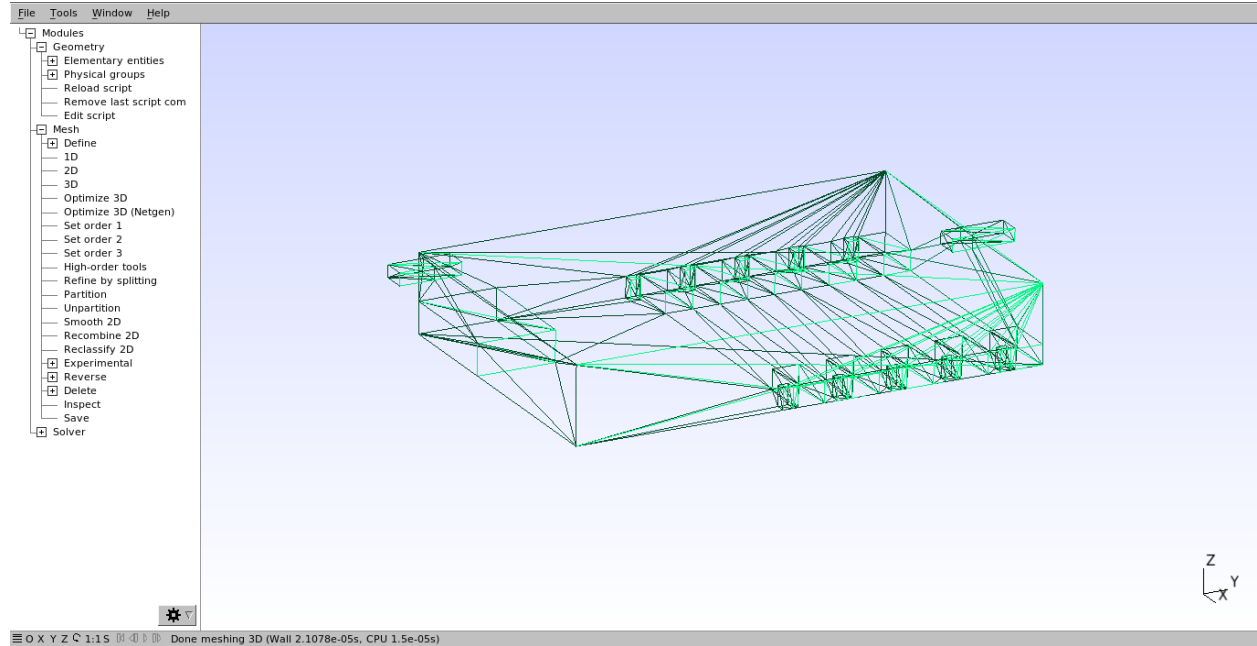


Figure 4.2 Mesh

4.4 Mathematical Modelling :

The temperature of the room is taken to be a constant 293 K. All the thermophysical properties are evaluated at this temperature.

Properties of air :

Temperature $T = 20^\circ \text{C}$

Kinematic viscosity $\nu = 1.6 * 10^{-6} \text{ m}^2/\text{s}$

Density $\rho = 1.204 \text{ kg / m}^3$

Continuity equation :

The differential form of continuity equation is,

$$\frac{\partial \rho}{\partial t} + \vec{\nabla} \cdot (\rho \vec{u}) = 0$$

Momentum equation :

The convective form of navier stokes equation is given by,

$$\rho \left[\frac{\partial \vec{v}}{\partial t} + \vec{v} \cdot \nabla \vec{v} \right] = -\nabla p + \nabla \cdot \bar{\tau} + \rho \vec{f}$$

Where \vec{v} is the flow velocity, ρ is the density, p is the pressure, τ is the stress tensor, \vec{f} represents the accelerations of the body acting on the continuum.

The general transport equation :

To solve the Navier-Stokes equation using the finite volume method, it is written in a particular form called the transport equation.

$$\frac{\partial(\rho\phi)}{\partial t} + \nabla \cdot (\rho\phi \vec{u}) = \nabla \cdot (\Gamma \nabla \phi) + S_\phi$$

Where ϕ is any general variable like temperature, u, v, w that varies along the flow field, Γ is the diffusion constant, ρ is the density. By setting the value of ϕ to u, v, w, T and selecting appropriate values of Γ and source terms we can get the mass, momentum and energy equations.

By integrating throughout the control volume,

$$\int_V \frac{\partial(\rho\phi)}{\partial t} dV + \int_V \nabla \cdot (\rho\phi \vec{u}) dV = \int_V \nabla \cdot (\Gamma \nabla \phi) dV + \int_V S_\phi dV$$

Temporal derivative

Convective term

Diffusion term

Source term

The entire solution domain is divided into a finite number of control volumes or cells and are discretized to produce a system of algebraic equations of the form

$$\begin{pmatrix} a_{11} & \cdots & a_{1n} \\ \vdots & \ddots & \vdots \\ a_{m1} & \cdots & a_{mn} \end{pmatrix} \begin{pmatrix} x_1 \\ \vdots \\ x_n \end{pmatrix} = \begin{pmatrix} b_1 \\ \vdots \\ b_n \end{pmatrix}$$

or,

$$\mathbf{Ax} = \mathbf{b}$$

Discretization :

The terms A and b in the matrix equation, are found out by the various discretization schemes used in the Finite Volume Method. Table 4.2 shows the advantages and disadvantages of each scheme used in OpenFOAM along with their application.

TERM	SCHEME	ACCURACY	SOLUTION TYPE	APPLICATION
Convection and Diffusion	Linear	2 nd order	Oscillatory / Unbounded	Diffusion
	Upwind	1 st order	Bounded and diffusive	Convective
	Linear upwind	2 nd order	Oscillatory	Highly convective with strong gradients
	Vanleer (TVD)	2 nd order	Bounded	
	LUST (TVD)	2 nd order	Blended 75% linear, 25% linear upwind	Convection and diffusion
Gradient ∇	Gauss linear	2 nd order	Good mesh quality	
	Least squares		Oscillatory on Tetrahedral meshes	
Time	steadyState	-		Steady state flows
	Euler	1 st order	Bounded, Stable and diffusive	Unsteady flows
	Backward	2 nd order	Accurate , unbounded	
	Crank Nicholson	2 nd order	bounded/ unbounded	
Laplacian	Orthogonal	2 nd order		Hexahedral meshes, no grading
	Corrected	2 nd order	Unbounded solutions	Non orthogonal, Orthogonal mesh with stretching
	Limited	2 nd order	Bounded / Unbounded	Non orthogonal,

				Unstructured meshes
	Uncorrected	2 nd order	Stable, more diffusive than limited and corrected, less accurate	Hexahedral meshes with very low non orthogonality

Table 4.2 Finite volume discretization schemes

4.5 Openfoam Case setup

‘0’ folder :

This folder consists of two files named ‘p’ and ‘U’ which represent initial pressure and velocity respectively. In OpenFOAM, pressure is defined in terms of pressure/density called kinematic pressure. Hence, the processed results have to be multiplied by density to get the correct value of pressure. The initial and boundary conditions are specified in these two files.

Surface	Boundary condition	
	U (m/s)	P (m ² /s ²)
Internal field	Uniform 0	Uniform 0
Main inlet	(-5.186, 0, 0)	Zero Gradient
Branch inlet	(0, 0, -5.186)	Zero Gradient
Main outlet	Zero Gradient	Uniform 0
Branch outlet	Zero Gradient	Uniform 0
Walls	No slip	Zero Gradient
Surfaces	No slip	Zero Gradient

Table 4.3 Initial and Boundary conditions

Constant :

This folder contains the details of geometry, mesh of the room and also the properties of air in the “transportProperties” file.

System :

This folder contains the

- control dictionary which contains the timestep and solver settings
- The fv schemes dictionary that contains the numerical schemes for various terms, such as convection, diffusion, time derivative etc.
- The fv solution dictionary which controls the tolerances, algorithms to solve pressure velocity coupling and equations.
- decomposePar dictionary that allows the user to run a simulation in parallel by decomposing the geometry into a number of separate entities.

Scheme selection :

TERM	SCHEME
Time	Euler
Gradient	Gauss linear
Divergence	Gauss limited linear
Laplacian	Gauss linear corrected
Interpolation	Linear upwind

Table 4.4 Schemes used in OpenFoam

Solver selection :

The solver used is “icoFoam”. This is the solver used for laminar, incompressible and unsteady flow. The “PISO” algorithm is used to solve pressure - velocity coupling in unsteady flows.

4.6 Post Processing :

After the OpenFOAM case is simulated, we visualise the output in ParaView. The performance of a ventilation system is quantified by studying the streamlines followed by air particles originating from the defined inlets. To achieve this, we define a point-source for the streamlines to originate from, in each inlet duct. The sphere associated with the point source has its radius adjusted so as to encompass the entirety of the inlet. This is done to ensure that the sample space of points of origination for the streamlines is as comprehensive as possible.

The streamlines are plotted using the velocity values generated from the simulation. The velocity value of the last timestep from the simulation is taken for this, to ensure that the simulation reaches steady state.

CHAPTER 5

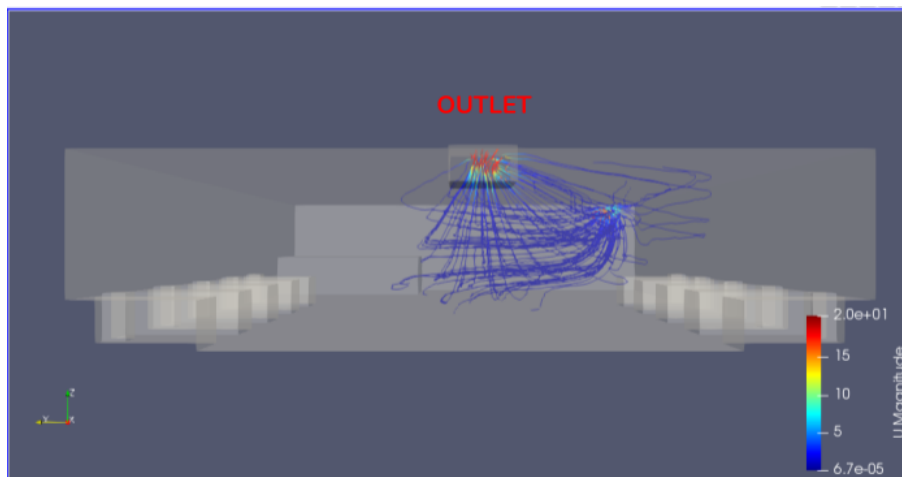
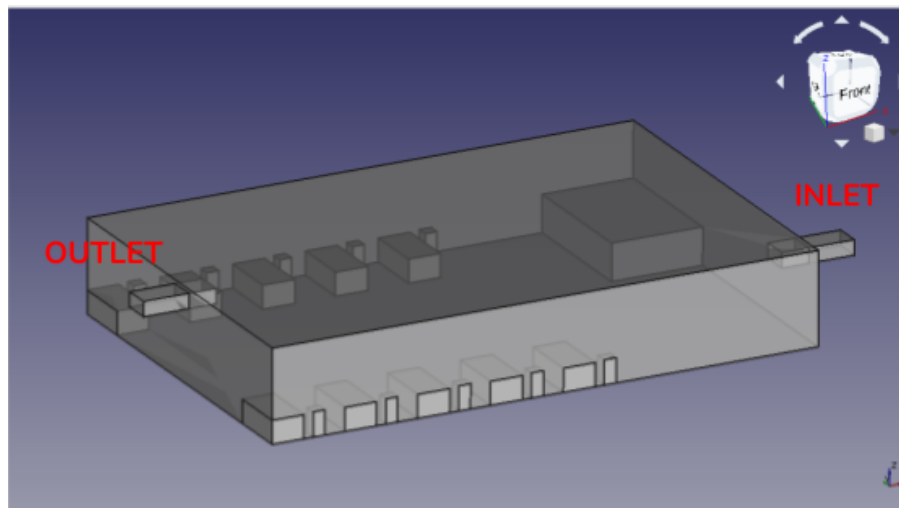
RESULTS AND DISCUSSION

The experiments are conducted with the aim of ensuring airflow throughout the room. For this, multiple case studies are performed with different inlet and outlet positions and the most suitable position is found so that there are less chances of infection to doctors, nurses and other healthcare workers. In this section, we discuss the following 4 designs :

- **Case - 1 : 1_inlet_1_outlet (Validation case)**
- **Case - 2 : 2_branch_pair_shifted**
- **Case 3 : 3_wall_4_ceiling**
- **Case 4 : 3_wall_5_ceiling**

5.1 1_inlet_1_outlet :

This is a test case that we used to check if our pipeline works as intended. Results show that the airflow is vacant in most parts of the room.



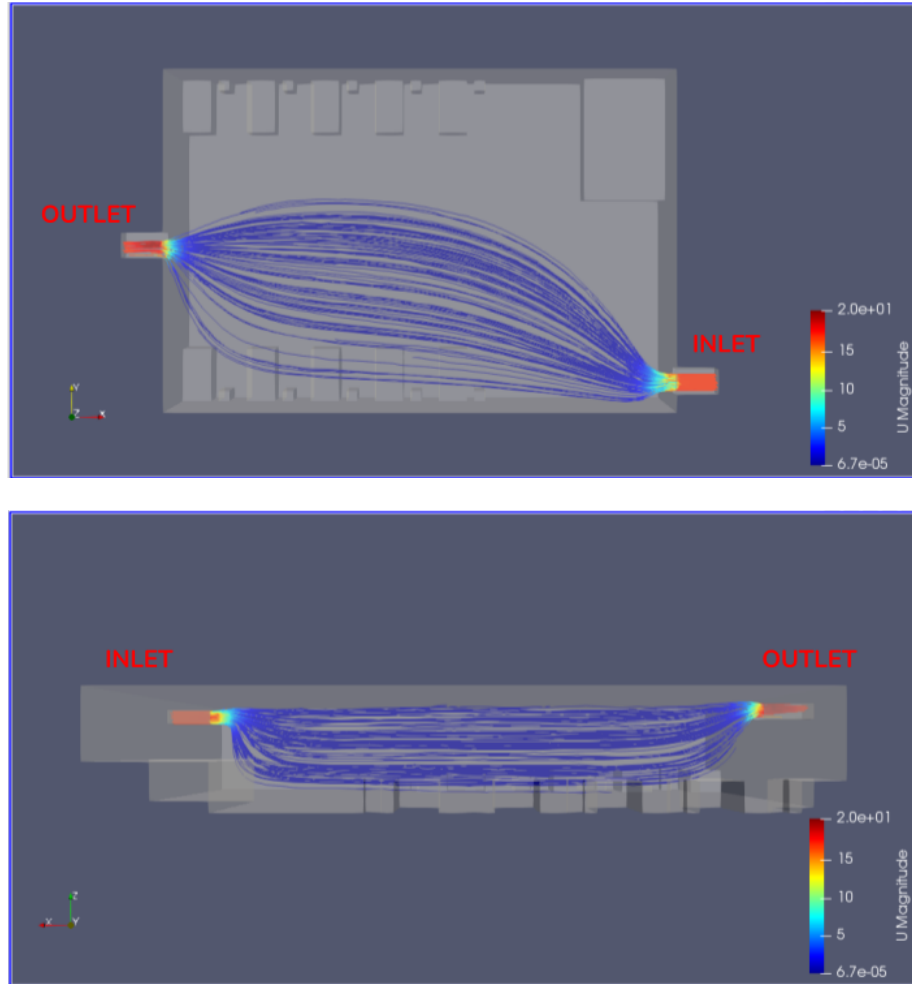
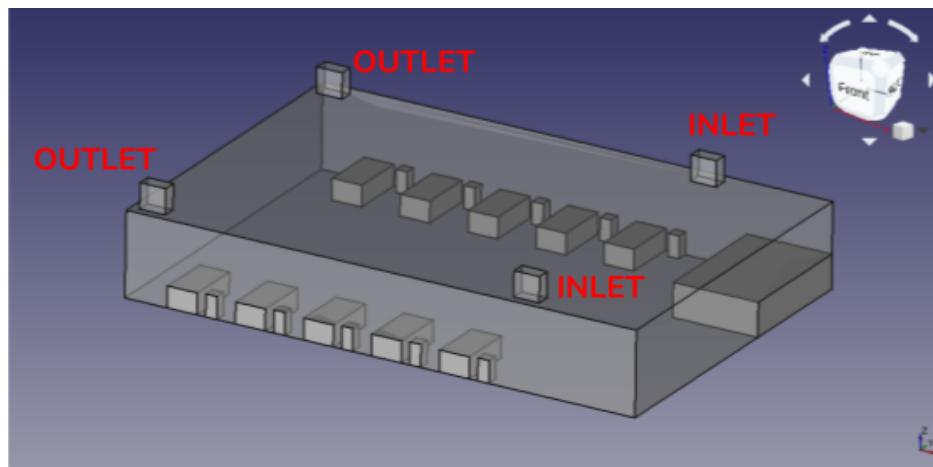


Fig 5.1 1_inlet_1_outlet

5.2.2 branch_pair_shifted :

The beds are shifted down the room by 5 feet and the main ducts are replaced with two pairs of ceiling mounted ducts running across the room. The results show improvement in air circulation in an overall sense, but the nurse station does not get enough ventilation.



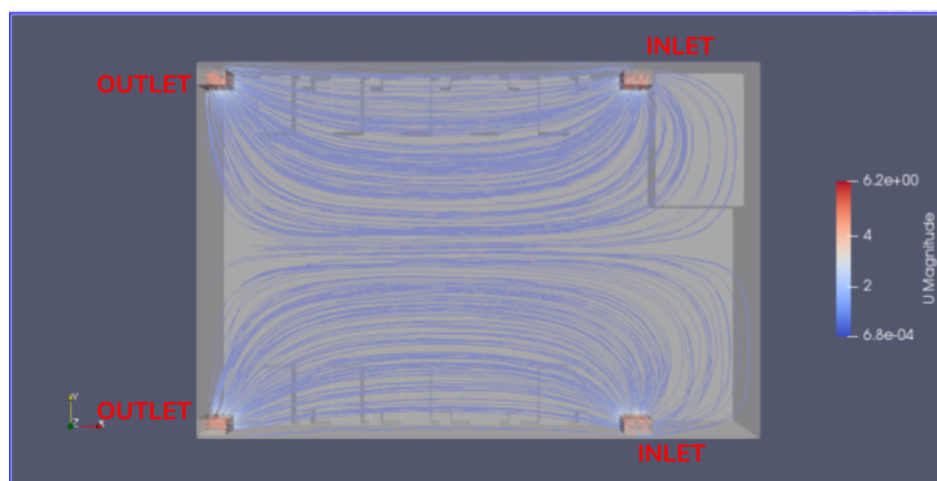
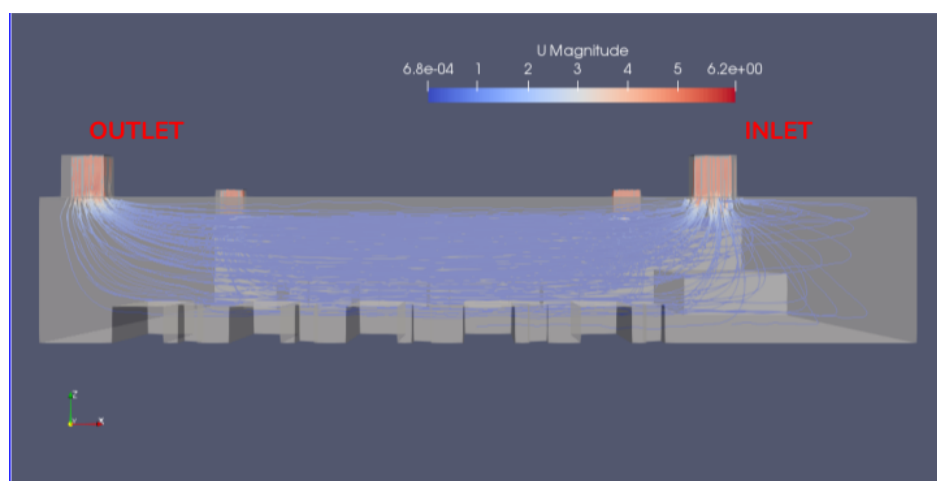
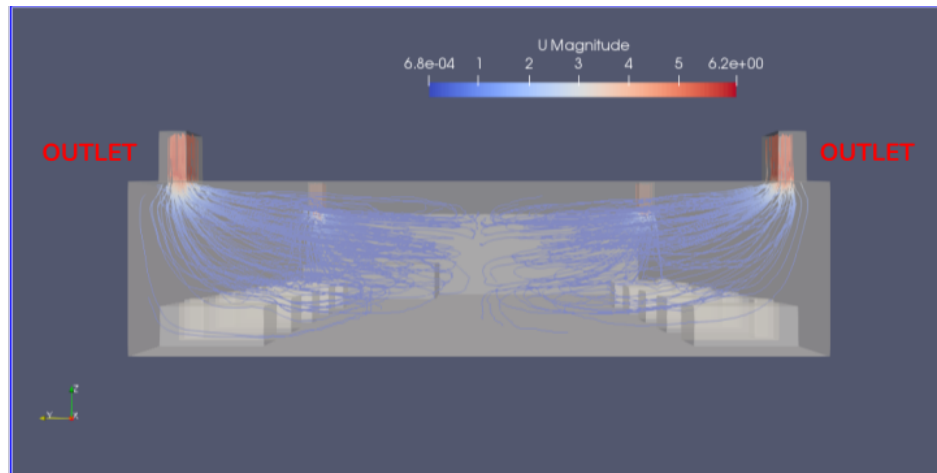
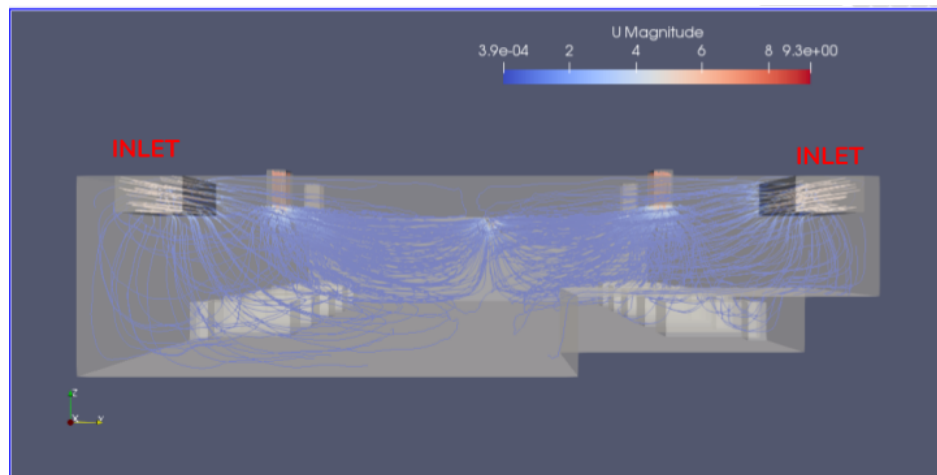
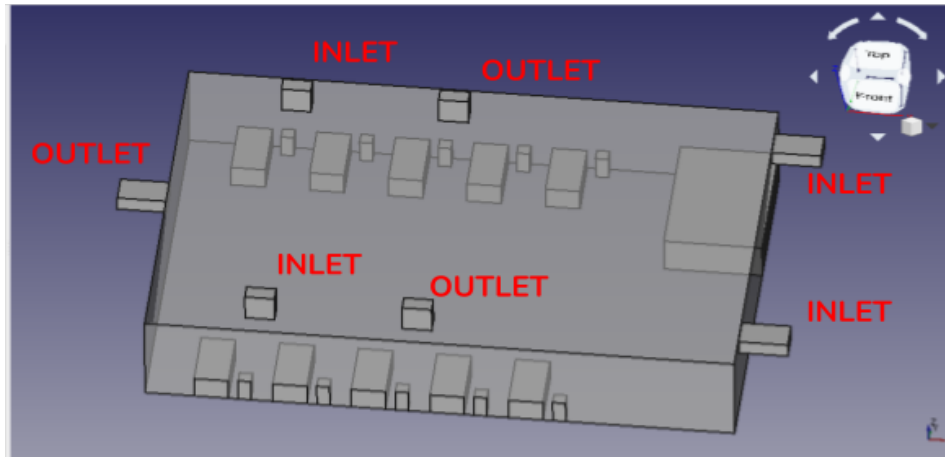


Fig 5.2 branch_pair_shifted

5.3.3 wall_4_ceiling :

Two wall mounted vents are added in an attempt to ensure air circulation around the nurse station. But a new problem arises in the form of a vacant region in the middle of the room where air does not flow properly.



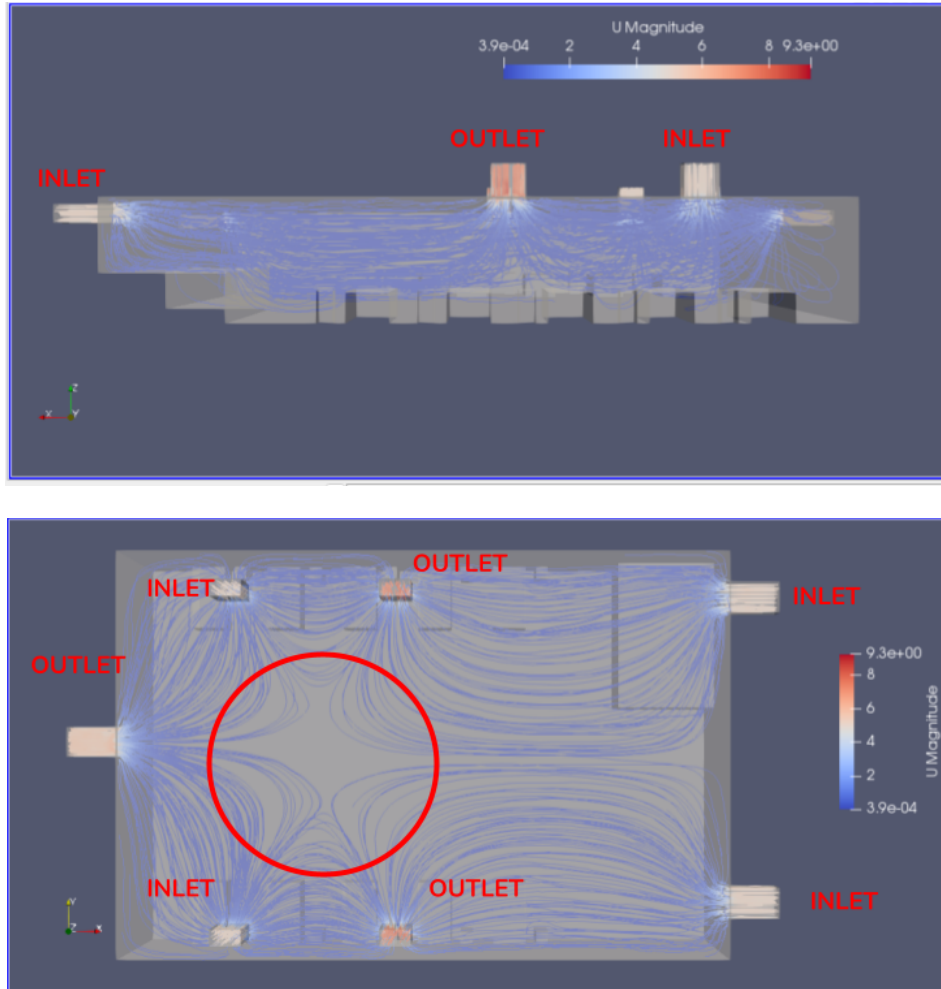
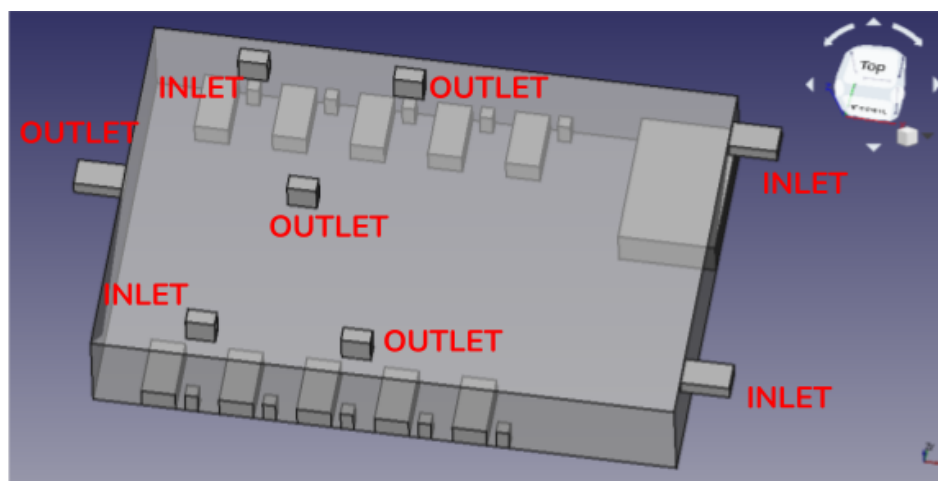


Fig 5.3 3_wall_4_ceiling

5.4 3_wall_5_ceiling :

An extra vent is added to the previous case to improve airflow in the vacant region. Results show the improvement in air circulation throughout the room.



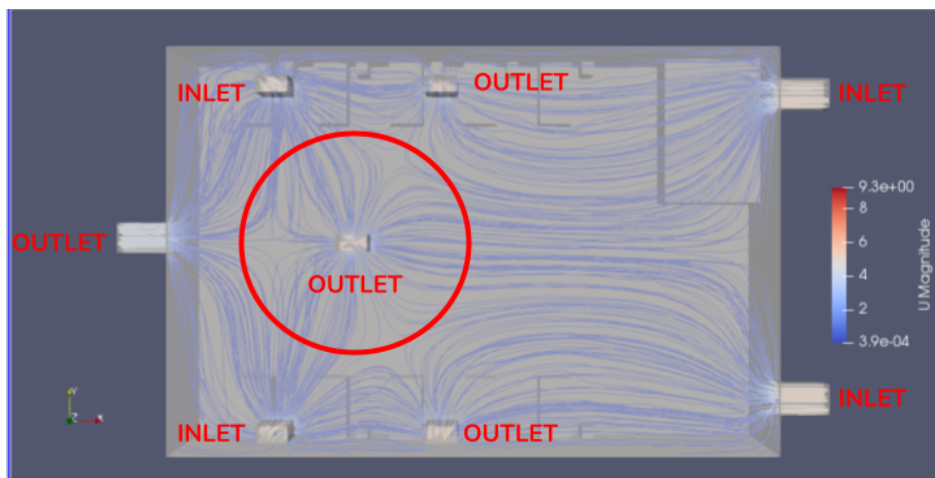
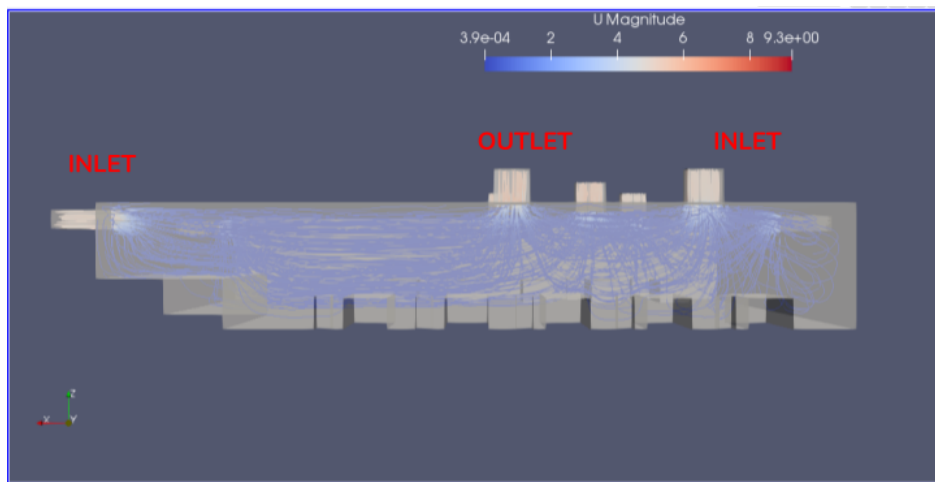
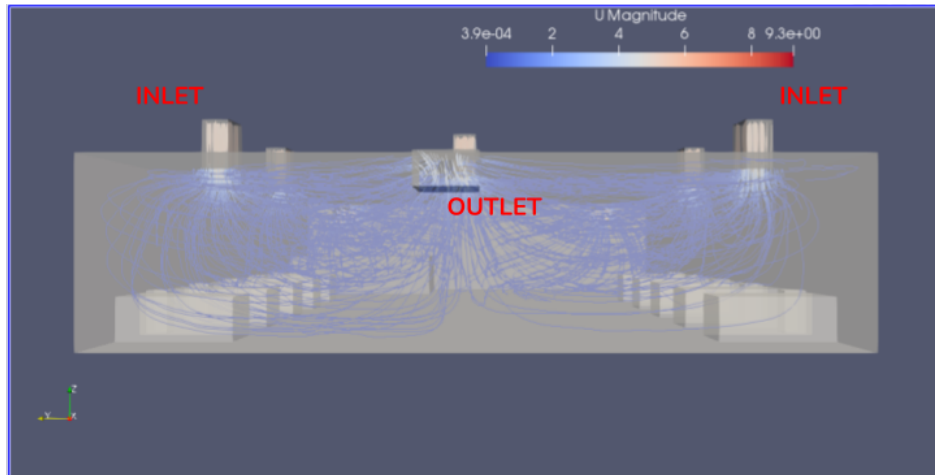


Fig 5.4 3_wall_5_ceiling

Comparing this result with the previous case, we can see that the flow in the choke point has been improved by the addition of an extra outlet as follows:

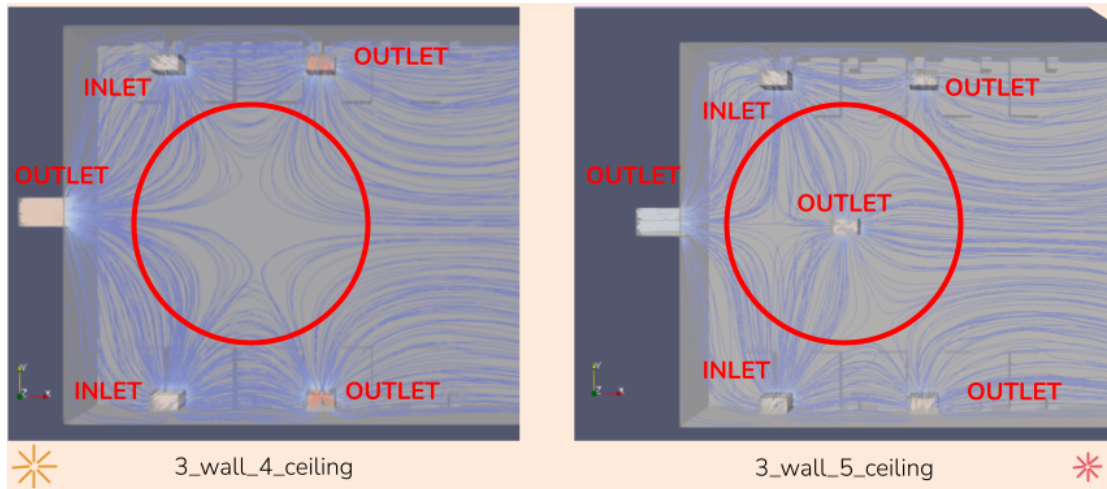


Fig 5.5 Comparison

CHAPTER 6

CONCLUSIONS AND FUTURE WORK

From the iterative design process, we can infer that the vent and bed positioning in the room from the “3_wall_5_ceiling” case appears to perform better than the other configurations studied. The streamlines plotted in this case seem to cover the entire room effectively. The nurse substation receives adequate ventilation, and there is almost no large swathe that is improperly ventilated.

Given our preliminary findings, we can work on addressing the following shortcomings in subsequent works:

- We made use of only laminar solvers in our project, but ventilation is a turbulent process. By applying the appropriate turbulence model, we should be able to further validate our inferences.
- We can also discuss our findings with microbiologists and healthcare professionals . Combined with their insights, we can further improve our design’s practicality in the real world.
- Our designs have been generated without accounting for the financial costs incurred during implementation. Further research is required to verify the cost effectiveness of the design.

We have released our CFD codes on GitHub for greater transparency, and have provided instructions for running our simulations and reproducing our results. The link to the same can be found in the references section.

CHAPTER 7

REFERENCES

[1] Application of open-source CFD software to the indoor airflow simulation

Wang, C., Sadrizadeh, S., Holmberg, H., 38th AIVC Conference, 2017

[2] Thermo-ventilation study by OpenFOAM

Limane, A., Fellouah, H., Galanis, N., Build. Simul. 8, 2015

[3] Natural Ventilation for agriculture buildings

Hong, S., Exadaktylos, V., Lee, I., Amon, T., Youssef, A., Norton, T., Berckmans, D., Computers and Electronics in Agriculture, Volume 138, 2017

[4] Comparison of OpenFOAM and ANSYS Fluent

Welahettige, P., Vaagsaether, K., 9th EUROSIM Conference, 2016

[5] OpenFOAM simulation of the natural ventilation system in a university chemical laboratory

Córdova-Suárez, M., Tene-Salazar, O., Tigre-Ortega, F., et al., E3S Web Conf. Volume 167, 2020

[6] Optimisation of Hospital Room by means of CFD for more efficient ventilation

Méndez, C., San José J.F., Villafruela J.M., Castro, F., Energy and Buildings, Volume 40, Issue 5, 2008

[7] The ventilation of multiple-bed hospital wards: Review and analysis

Beggs, C., Kerr, K., Noakes, C., Hathway, A., Sleight, A., American Journal of Infection Control, 2008

[8] Dynamic airflow simulation within an isolation room

Shih, Y., Chiu, C., Wang, O., Building and Environment, Volume 42, Issue 9, 2007

[9] Development of ventilation design strategy for effective removal of pollutant in the isolation room of a hospital

K.W.D. Cheong, S.Y. Phua, Building and Environment, Volume 41, Issue 9, 2006

[10] Numerical Study of Three Ventilation Strategies in a prefabricated COVID-19 inpatient ward

Juan Ren, Yue Wang, Qibo Liu, Yu Liu, Building and Environment, Volume 188, Issue 9, 2007

[11] Role of ventilation in airborne transmission of infectious agents in the built environment – a multidisciplinary systematic review

Y. Li, G. M. Leung, J. W. Tang, X. Yang, et al., International Journal of Indoor Environment and Health, 2007

[12] Virions and respiratory droplets in air: Diffusion, drift, and contact with the epithelium

Vladimir P. Zhdanov, Bengt Kasemo, Biosystems, Volume 198, 2020

[13] A physicist view of COVID-19 airborne infection through convective airflow in indoor spaces

Luis A. Anchordoqui and Eugene M. Chudnovsky, ArXiv, 2020

NCDC Guidelines for Setting up Isolation Facility/Ward

<https://ncdc.gov.in/WriteReadData/l892s/42417646181584529159.pdf>

Last Accessed : 22nd December, 2021

BIS 659 (1964) : Safety Code for Air Conditioning

<https://law.resource.org/pub/in/bis/S08/is.659.1964.pdf>

Last Accessed : 22nd December, 2021

HVAC - How to Size and Design Ducts

<https://www.cedengineering.com/userfiles/HVAC%20-%20How%20to%20Size%20and%20Design%20Ducts%20R1.pdf>

Last Accessed : 22nd December, 2021

Hospital Bed Dimensions

<https://www.indiamart.com/proddetail/hospital-bed-12755997173.html>

Last Accessed : 22nd December, 2021

Parallel Computing with OpenFOAM

<http://www.wolfdynamics.com/wiki/parallel.pdf>

Last Accessed : 22nd December, 2021

Air Cleaning for COVID 19

<https://www.epa.gov/coronavirus/air-cleaners-hvac-filters-and-coronavirus-covid-19>

Last Accessed : 22nd December, 2021

GitHub Repo

<https://github.com/Naimish240/CFD-For-Ventilation-Analysis>

Last Accessed : 22nd December, 2021