FOSSEE, IIT Bombay
OpenFOAM Case Study Project
22 April, 2022

Effect of Ceiling Fan on the Ventilation of Nuclear Lab, IIT Bombay

Soham Sachin Purohit
Indian Institute of Technology Bombay

April 2022

Abstract

A previous ventilation analysis of the Nuclear Lab, IIT Bombay revealed poor ventilation conditions which did not meet the safety requirements for preventing COVID transmission. Solutions in the form of installation of exhaust fans at optimum locations along with open door/window configurations were proposed. This project extended this study to the use of a ceiling fan as a cost-effective alternative and performed a comparison with the previous study. The results obtained showed comparable results to the previous study.

1 Introduction

Due to the COVID-19 pandemic, the government had mandatorily closed all classrooms, laboratories, offices, and other workspaces. As restrictions began to ease, IIT Bombay began seeking ventilation analyses of several such rooms, and solutions to ensure that the safety requirements with respect to COVID were being met. One such study was done for the Nuclear Lab, IIT Bombay, which revealed poor ventilation conditions. This study proposed the installation of an exhaust fan at an optimal location along with a requirement of keeping the door and the window in the room open while operating the two air conditioners present in the room at a sufficiently good capacity. The results proved significantly better ventilation than before, which met safety requirements. However, there were high costs regarding the operation of the air conditioners while keeping the door and window open, which made it a cost-inefficient solution. Through this study, our objective is to explore the installation of a ceiling fan and study the effect on ventilation as a direct comparison to a previous study, as a cost-effective alternative. The installation, maintenance, as well as operation of a ceiling fan is significantly cheaper and would allow the use of the door and the window to be kept open without any added costs. The results obtained were better than a few cases from the previous study and were comparable to the best case obtained from that study.

(a) Front view of wall with window and ACs (b) Front view of wall with door

Figure 1: Nuclear Lab, IIT Bombay

2 Problem Statement

Figure 1 displays images of the Nuclear Lab. We can see the two ACs, and window in the first image, and the door in the second. A CFD analysis was performed for this geometry with operational ACs using Ansys 2021 R1. We now wish to have a simulation with a ceiling fan present in the room and the ACs switched off, in order to validate the installation of this fan as a potential cost effective alternative to the solution proposed in the previous study.

3 Governing Equations and Models

Navier Stokes equations for single phase flows are the governing equations for the fluid flow, which are then followed by the k-ω SST 2-equation based model that captures turbulence in the flow. The Navier Stokes Equation is as follows:

$$\nabla \cdot u = 0 \tag{1}$$

$$\nabla \cdot (u) - \nabla \cdot R = -\nabla p + S_u \tag{2}$$

The SST k-omega turbulence model is a two-equation eddy-viscosity model that is used for many aerodynamic applications. It is a hybrid model combining the Wilcox k-omega and the k-epsilon models. A blending function, F1, activates the Wilcox model near the wall and the k-epsilon model in the free stream. This ensures that the appropriate model is utilized throughout the flow field. The k-omega model is well suited for simulating flow in the viscous sub-layer. The k-epsilon model is ideal for predicting flow behavior in regions away from the wall. The equations for the k-ω SST models are as follows:

$$\frac{\partial}{\partial t}(\rho\omega) = \nabla \cdot (\rho D_\omega \nabla\omega) + \frac{\rho\gamma G}{\nu} - \frac{2}{3}\rho\gamma\omega(\nabla \cdot u) - \rho\beta\omega^2 - \rho(F_1)CD_\kappa\omega + S_\omega \tag{3}$$

$$\frac{\partial}{\partial t}(\rho\kappa) = \nabla \cdot (\rho D_\kappa \nabla\kappa) + \rho G - \frac{2}{3}\rho(\nabla \cdot u)\kappa - \rho\beta * \omega\kappa + S_\kappa \tag{4}$$

$$\nu_t = a_1 \frac{\kappa}{max(a_1\omega, b_1 F_{23} S} \tag{5}$$

4 Simulation Procedure

4.1 Geometry and Mesh

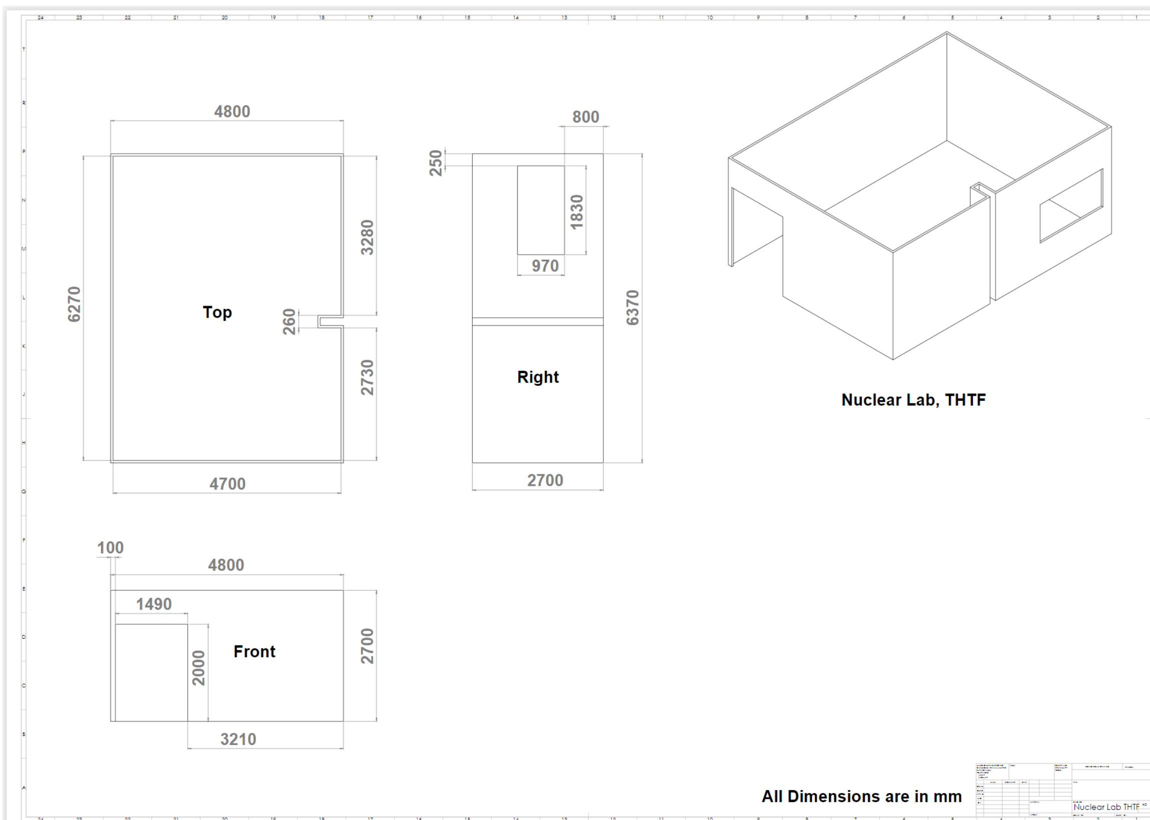
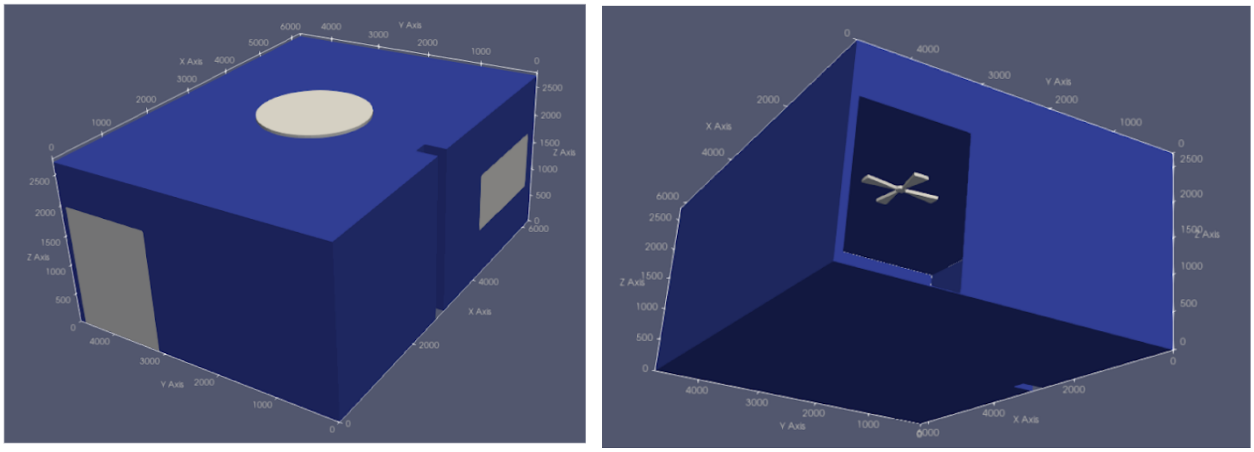


Figure 2: Dimensions of the Nuclear Lab, IIT Bombay

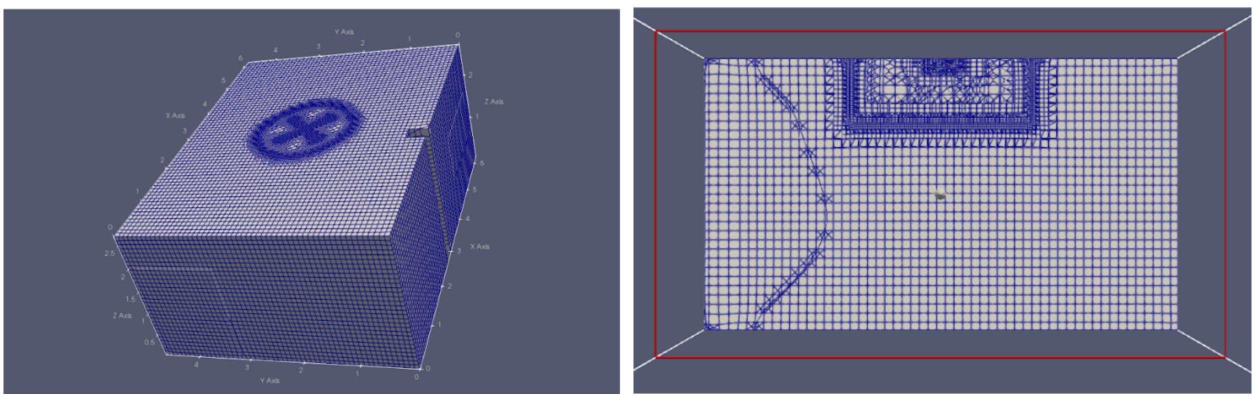
The geometry was prepared using the exact dimensions of the room, as shown in Figure 2. Solidworks 2020 was used in generating the individual STL files for the room, door, window, fan and AMI region. The AMI region consisted of a cylinder coaxial with the fan, which is used to define the cells which would be a part of the Rotating Cell Zone. The dimensions of the fan were those of a standard commercial fan suitable for the dimensions of the room. The final geometry is displayed in Figure 3, where the ceiling fan is clearly visible.

BlockMesh was used to construct a coarse mesh, with (65, 55, 30) cells generated along the 6.37m 4.7m 2.7m dimensions of the room respectively. Expansion ratios were kept as 1 in all directions. Finer mesh was generated for the fan and AMI (rotating cell zone) using



(a) Central white cylinder is AMI region (b) Fan placed at the center of the room

Figure 3: Final generated geometry of the room



(a) External view of the mesh (b) Slice through the center

Figure 4: Contrast between blockMesh and snappyHexMesh is clearly visible

snappyHexMesh, with a level 2 refinement for both. Again, expansion ratios were kept at 1. The generated mesh is displayed in Figure 4.

4.2 Initial and Boundary Conditions

The initial and boundary conditions are as follows:

- **Velocity:** door- velocity inlet with 2 m/s air flowing inwards; window- pressure inlet-outlet velocity boundary condition; fan- moving wall velocity with 0 m/s; walls- no slip boundary condition; AMI- cyclic AMI boundary condition
- **Pressure:** door- fixed pressure flux with 0 gradient and 0 value; window- pressure outlet with value 0; fan- fixed pressure flux with 0 gradient and 0 value; walls- fixed pressure flux with 0 gradient and 0 value; AMI- cyclic AMI boundary condition
- **Turbulent Kinetic Energy (k):** door- turbulent intensity kinetic energy inlet;

window- zero gradient; fan- kqr wall function; walls- kqr wall function; AMI- cyclic AMI boundary condition

- **Specific Rate of Dissipation (ω):** door- turbulent mixing length frequency inlet; window- zero gradient; fan- omega wall function; walls- omega wall function; AMI- cyclic AMI boundary condition
- **Rotating Cell Zone:** Angular velocity = 36.65 rad/s= 350 rpm (commercial fan)

4.3 Solver

Our main interest is in studying the steady state behaviour of air and checking for any re-circulation or stagnation zones that may be present, we want to have a solver that can perform steady state, turbulent simulations. For this, we use the pimpleFoam solver. PimpleFoam is a large time-step transient solver for incompressible flow using the PIMPLE (merged PISO-SIMPLE) algorithm.

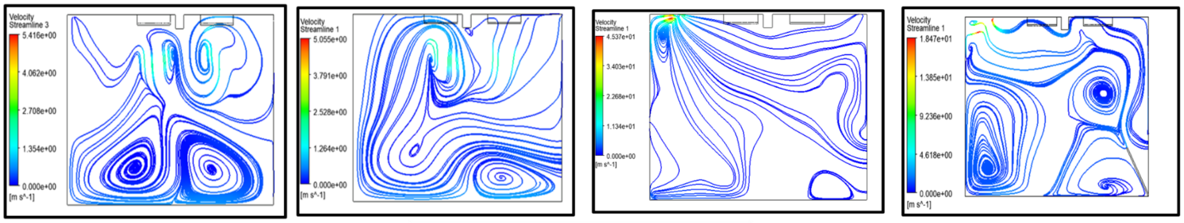
We use k-ω SST model for turbulence, to ensure that we get accurate solutions both close to the wall as well as within the domain. The values are calculated taking into account the standard y+ values of 80 and the characteristic length of 1m. We keep an end time of 1s, with an end time of 0.0002s since we only want the steady state results, which would not be affected by a small end time.

5 Results and Discussions

Our primary objective was to have a comparison with the previous study. As mentioned already, the previous study performed the analysis for various cases. The contour plots of velocity streamlines for 1.5 m height and 1 m height from the floor were generated and analysed. These were our primary heights of interest since our intention is to check for re-circulation and stagnation in the context of COVID particles, which would be transmitted through the respiratory tracts of humans. Since sitting and standing are the most common situations, we take the heights (1.5 m and 1 m respectively) at which the nose and mouth of a person in the room would be present.

The results of the four cases for 1.5 m height are displayed in Figure 5. Case 1, which had both door and window closed, has the highest number of recirculation zones and very poor ventilation. Case 3 is the best case with fully open door and window, and an operating exhaust fan at the top right corner, near the window.

We observe that there is a large recirculation zone with almost stagnant air is present in the bottom left corner of the room in the first images of both Figure 6 and 7. This is caused due to a lack of an outlet for air to escape. Another small recirculation zone is present in the top right corner of both the images. Apart from this, the rest of the room does not have stagnant air, and a direct pathway exists for air to freely flow out of the room through



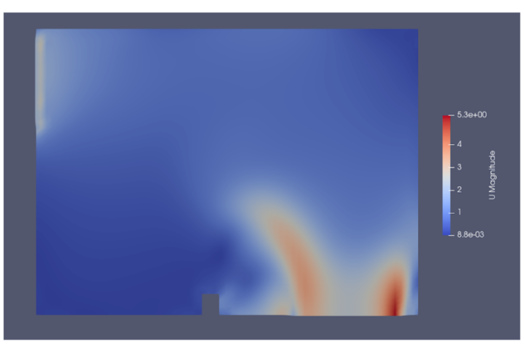
Case	Window	Door	Exhaust Fan
1	Closed	Closed	Off
2	Open	Open	Off
3	Open	Open	On
4	Open	Partially Open	On

Figure 5: Velocity streamline contours at 1.5m height for various cases, from previous study

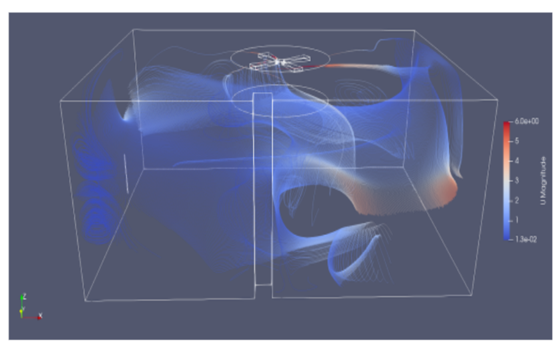
either the door or the window. Comparing directly with the previous study, we can draw the following inferences:

- Significantly better situation than Case 1, and slightly better situation than case 2 of the previous study. This is because the fan creates greater speeds of air flow than ACs.
- Case 3 of previous study is still the best. This is because the AC present on the wall close to the corner acts as an outlet to eliminate the recirculation zone present in that corner, which still exists for our case.
- Keeping the above issue in mind, we can explore one out of the following three possibilities, which would still be significantly more cost efficient than operating two ACs with open door and window- 1) Install an exhaust fan in the recirculation zone region, or 3) have an oscillating floor fan present in that corner to divert air away, or 2) operate the single AC present just above that region, as an outlet.

The results of the simulation are now displayed for 1.5 m (Figure 6) and 1 m (Figure 7)

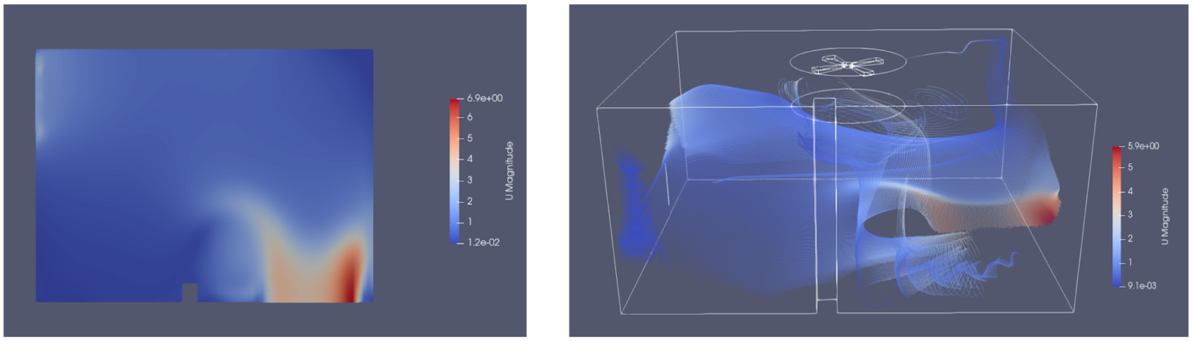


Velocity Contours at 1.5m height



Streamlines that have the diagonal of the slice at 1.5m height as the seed line

Figure 6: Simulation Results at 1.5m height (current study)



Velocity Contours at 1m height

Streamlines that have the diagonal of the slice at 1m height as the seed line

Figure 7: Simulation Results at 1m height (current study)

In summary, the results of the simulation proved that the usage of a ceiling fan would solve a large number of issues present in the current state of ventilation in the room (Case 1 of previous study). However, the usage of the ceiling fan alone does not make our ventilation sufficiently good enough for it to exceed our safety requirements, and one more modification from the above three options needs to be done in order to meet our requirements. Finally, this would be a cost saving alternative with comparably good results.

References

[1] Wilcox, D. C. (1998). Turbulence modeling for CFD. La Canada, Calif: DCW Industries.
[2] Nguyen, V.-B., Do, Q.-V., & Pham, V.-S. (2020). An OpenFOAM solver for multiphase and turbulent flow. In Physics of Fluids (Vol. 32, Issue 4, p. 043303). AIP Publishing.
[3] OpenFOAM Documentation for Rotating Mesh
[4] Turbulence Modeling in OpenFOAM
[5] Cyclic Arbitrary Mesh Interface (AMI) OpenFOAM Documentation