Study of Air Recirculation Zones in Shared Spaces

Dr. Gajendra Vasantrao Patil
Pillai HOC college of Engineering and Technology, University of Mumbai, Khalapur, HOC Colony Rd, Taluka, Rasayani, Maharashtra 410207, India
Email: gpatil@mes.ac.in

Varsha Pol
Research Scholar, Pillai HOC college of Engineering and Technology, University of Mumbai, Khalapur, HOC Colony Rd, Taluka, Rasayani, Maharashtra 410207, India
Email: varshapol3@gmail.com

Abstract:

The Air borne transmission is a very big concern for highly infectious diseases like Covid-19 and other airborne diseases. A micro droplet and aerosol can be carried out in the air and can remain flowing in air over a distance in a confined space, leading to affecting high number of people getting prone to infection and it is very dangerous in enclosed spaces or shared spaces. Public places, shared facilities are the areas, where infectious aerosol can be present in the air for a long duration. Ventilation of closed spaces, shared spaces is the need of hour to have analysed and deep study in context of infectious airborne diseases. Introduction of fresh air into the enclosed environment at regular interval of times may lead to fast dilution of air present in the enclosed space. The prominent building codes and HVAC guidelines allows as to calculate ACPH (Air changes per hour) in an enclosed space as per the occupancy and flow rate. The age of air is the criteria to define the amount of air residing in the enclosed space when it enters the space till its exhaust from that space. The more the age of air in the particular area the more can be the infection probability among the occupants. It is predominant to study the airflow pattern caused due to ventilation which can be collaborated with age of air to know about the infection probability. Typically, a classroom geometry is assumed with inlet outlet boundary conditions where exhaust fan is playing a major role of displacement ventilation. Study of air recirculation zones and dead zones is the point of interest of this study. Computational fluid dynamics is the most powerful tool in the present era to study the air flow pattern in enclosed and shared spaces

Keywords: **Ansys; Computational Fluid Dynamics; SolidWorks; Fluent; Airflow simulation; computer aided design**
DOINumber:10.14704/NQ.2022.20.8.NQ22993 **NeuroQuantology2022;20(8): 9756-9768**

1.Introduction

The spread of respiratory diseases via aerosol particles in indoor settings is of significant concern (M.R.R.S. van Beest, 2022). There were various studies done for outdoor environment but there is need for indoor air assessment too. The introduction of indoor air quality has opened up various technological advancements in the pandemic era. The importance of indoor air for breathing is now being a active field of study. Clean and helathy air is key to healthy life. Indoor environments are contributing to the potential risk of spreading an infectious disease, as the likelihood of infected people sharing the same air with other people is high (M.R.R.S. van Beest, 2022). Ventilation of shared indoor spaces is crucial for mitigating air-borne infection spread among its occupants. Replacing the air in a room with fresh air is key to minimize the concentration of potentially infectious aerosol generated in the room. Recirculating air flow present at corners and around obstacles can trap air and infectious aerosol. As the aerosol size is comparatively small than 0.25 microns so it can be fairly assumed that most of the particles will move along the airflow and as the size is small the buoyancy effects can also be neglected for this particular case.

The objective of this paper is to use CFD to study the airflow pattern in the classroom, and how the effect of windows opened and closed condition can affect the flow patterns and recirculation zones in the fluid domain. There is specifically a need to identify those recirculation zones so that accumulation of pathogens can be terminated from those zones.

2 Literature Review

After reviewing recent literature destratification of air in a closed room is a major point of study. The project aims at designing and performing analysis on Exhaust Fans to accumulate all forms of knowledge about airlow in enclosed spaces and age of air. Comparing the airflow of classroom with exhaust fans.

(canada, 2021)Serhiy Yarusevych University of Waterloo canada (21-7-2021), "Experimental investigation of indoor aerosol dispersion and accumulation in the context of COVID-19: Effects of masks and ventilation" In this paper, the quantitative insight into the effect of common face masks and ventilation/air purification is given, It also provide relevant experimental metrics for modeling and risk assessment. It explains high-efficiency masks, such as the KN95, still offer substantially higher apparent filtration efficiencies (60% and 46% for R95 and KN95 masks, respectively) than the more commonly used cloth (10%) and surgical masks (12%),relatively low air-change rates (2 times/h) lead to lower aerosol build-up compared to the best mask in an unventilated space.

(Mikko Auvinen, 2022) Mikko Auvinen Finnish Meteorological Institute Helsinki, Finland (01-01-22), "High-resolution large-eddy simulation of indoor turbulence and its effect on airborne transmission of respiratory pathogens—Model validation and infection probability analysis" This paper deals with study indoor air turbulence and its effect on the dispersion of respiratory virus-laden aerosols and subsequent transmission risks. In this experimental aerosol concentration measurements are carried out, and their results are used to successfully validate the LES model results. It says LES dispersion results are subjected to pathogen exposure and infection probability analysis in accordance with the Wells–Riley model. It explains Air purifiers leads to greater reduction in absolute risks compared to the analytical Wells–Riley model.

(Ashkan Davani, 2022) Ashkan Davani University of Southern California (02-03-2022) "Minimizing the COVID-19 spread in hospitals through optimization of ventilation systems" This paper explains about poorly designed ventilation system acts like a perfectly stirred reactor—which enormously increases the possibilities of contamination, when air is injected from the ceiling and extracted from behind the patient beds, the infection spread is least probable since the particles exit the room orders of magnitude faster.

From (Joe E. Madsen, 2011) and (Shultz & Williams., 2007) it can be investigated that how ceiling fan and HVLS fans provide Air movement in a large space using CFD analysis which results in finding the optimum position and height of installation, which helps in reducing the cost for testing specimen. Design of HVLS fan to improve the airflow in a room, Analysis of existing airflow of ceiling fan in a room, Analysis of airflow around the HVLS fans, Finding the optimum design for both HVLS and ceiling fan, to perform possible outcomes comparing the results and establishing the relation of using HVLS fans for better efficiency in an enclosed space.

3 Outline of CFD Method

An ideal classroom of size 10x10x4 meters was considered for the study. For the sake of simplicity and keeping the computational time in mind the only the fluid domain was modelled with inlet as door and outlet as exhaust fans. The inlet door and outlet exhaust fans are simply kept as faces in the geometry. The students and other objects are not modelled to seek less computational time and robust mesh for the problem. The geometry is created in SolidWorks and only the fluid domain is considered for the flow visualization.

Procedure of Analysis in Ansys:

1. Geometry creation
2. Meshing
3. Solution in fluent

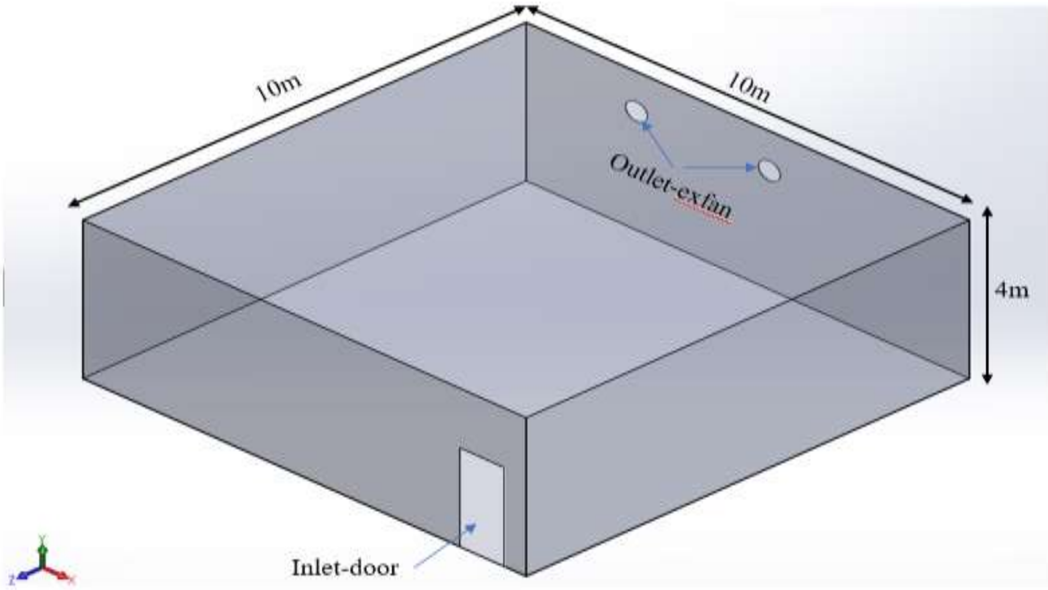
The geometry depicted below in figure1 is imported to Ansys for flow visualization. From analysis point of view the geometry is simple from computational point of view. Complex geometry will result in more computational power, higher number of elements and probability if more scope for errors.

The analysis has been done in two cases,

1. Both Exhaust Fans working without windows
2. Both Exhaust Fans working With Window open

3.1 Case 1: Both Exhaust Fans working without windows

In this study the effect of exhaust fans on without windows is visualised. Door size: 1 x 2.5 m, Exhaust Fan size: a circular plane of 0.5m diameter. For the sake of simplicity, the windows are not modelled as they are presumed to be completely closed. To observe and conclude the effects on flow due to exhaust fans in the enclosed facility. The flow rate of exhaust fans is 1500 cubic meter per hou r.



3. Figure 1: Ideal classroom without windows

3.1.1 Geometry Creation: the geometry of this case can be referred above in figure 1.

The following steps are to be followed in the modeller section of analysis, for better results the geometry should not be more complex so that while exporting it from any design software the multiple surface creation for the same part can be avoided. For Analysis of HVLS fan the geometry is first surrounded by the enclosures known as fluid domain under which the flow has to be calculated.

FLUID DOMAIN (ENCLOSURES): The fluid domains in our design is the room with shell i.e. the actual fluid domain is under the walls created. Enclosures can be of any shape depending upon the type of geometry, but the selected enclosure should be around the geometry enclosing it completely without any leakage or intersection between geometry and the enclosure. The room enclosure is basically the cuboid shape in which the actual fluid domain resides and which needs to be subtracted or extracted from the geometry.

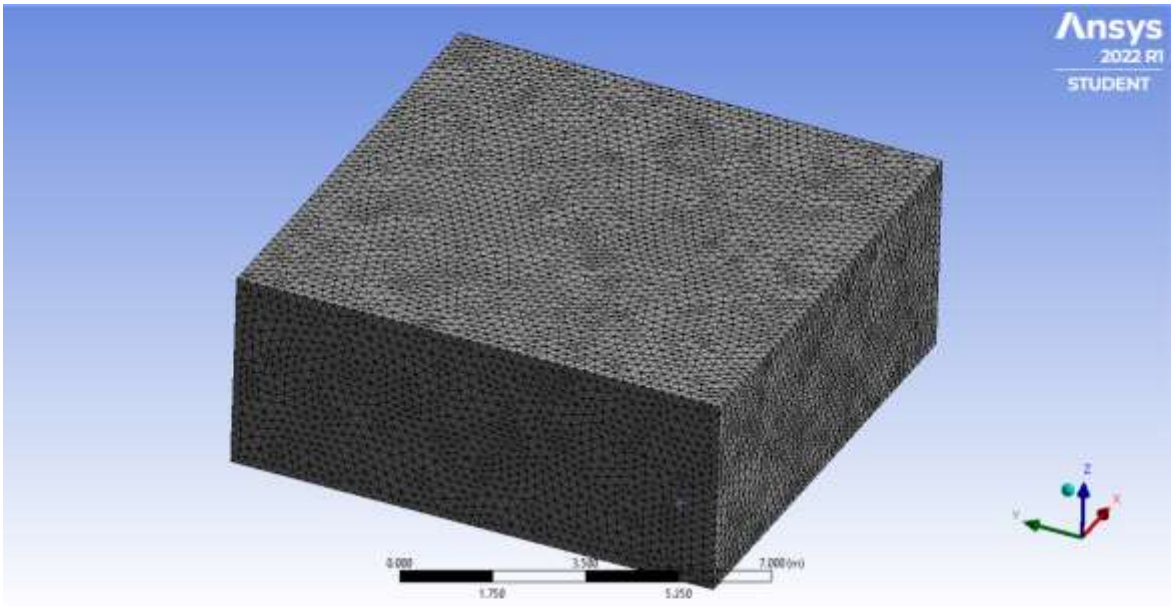
BOOLEAN (SUBTRACTION): Boolean operation are performed when the geometry is fully defined. It lets the software know about the contact regions in the domain and creating a cavity by subtracting the geometries. In our example the room being the cuboidal fluid domain will be subtracted from the inner domain.

NAME SELECTIONS: It is very important with respect to analysis as for the geometry to be user friendly its parts to be named properly, door will be named inlet-door, exhaust fans will be named outlet-exfans1 and outlet exfan 2 and for windows its inlet-windows 1,2and3 respectively. Naming can be done during meshing also.

3.1.2 Meshing:

Mesh independency test is done to reduce the dependency of geometry on number of cells, increasing number of cells does not conclude more precise results, it may also lead to high degree of error as the number of equations increase with the number of cells. So, balance between the number of finite elements and geometry size is to be maintained for robust design. Discretising the fluid domain into small cells to calculate the equations on each cell. The mesh created should be independent of the change in number of cells i.e., the solution should not change after the change in mesh. Unstructured tetrahedral mesh is used for the study. A line in the centre of the domain is considered on which the velocity magnitude is plotted against the y axis.

With Numerous cell sizes of 0.5 to 0.1m, it was found that the solution of 0.1 and 0.2m is quite similar. To minimize the computational time we will consider cell size of 0.2m and cell count of 1.7k.

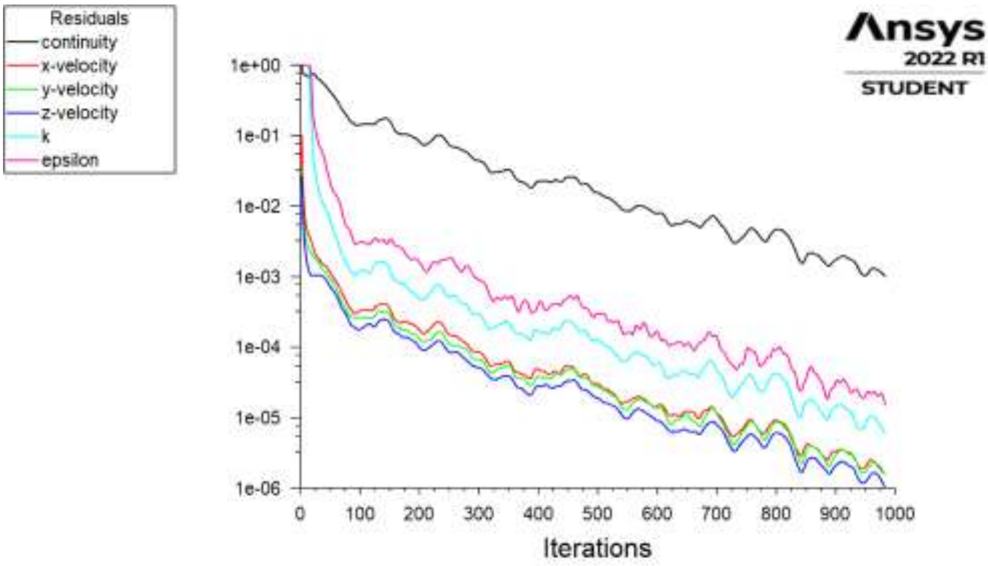


4. Figure 2: Meshing for classroom without windows

3.1.3 Results:

In all the cases the k-epsilon turbulence model with steady state condition will be used, the domain is considered as fluid domain. The inlet is defined as pressure inlet and the outlet as velocity outlet. The walls are stationary with no-slip condition. Hybrid Initialization is done with residuals kept below 10^{-3}. The residuals for continuity fall below 10^{-3} and for turbulent kinetic energy, dissipation rate has an falling nature residuals below 10^{-4}, which is a good sign of robust solution.

NEUROQUANTOLOGY | AUGUST 2022 | VOLUME 20 | ISSUE 8 | PAGE 9756-9768| DOI: 10.14704/NQ.2022.20.8.NQ22993



5. Figure 3: Residual for the case 1

From the above it is evident that the residuals have a falling nature and after 1000 of iteration the residuals are still falling without any much noise in the curves.

Results:

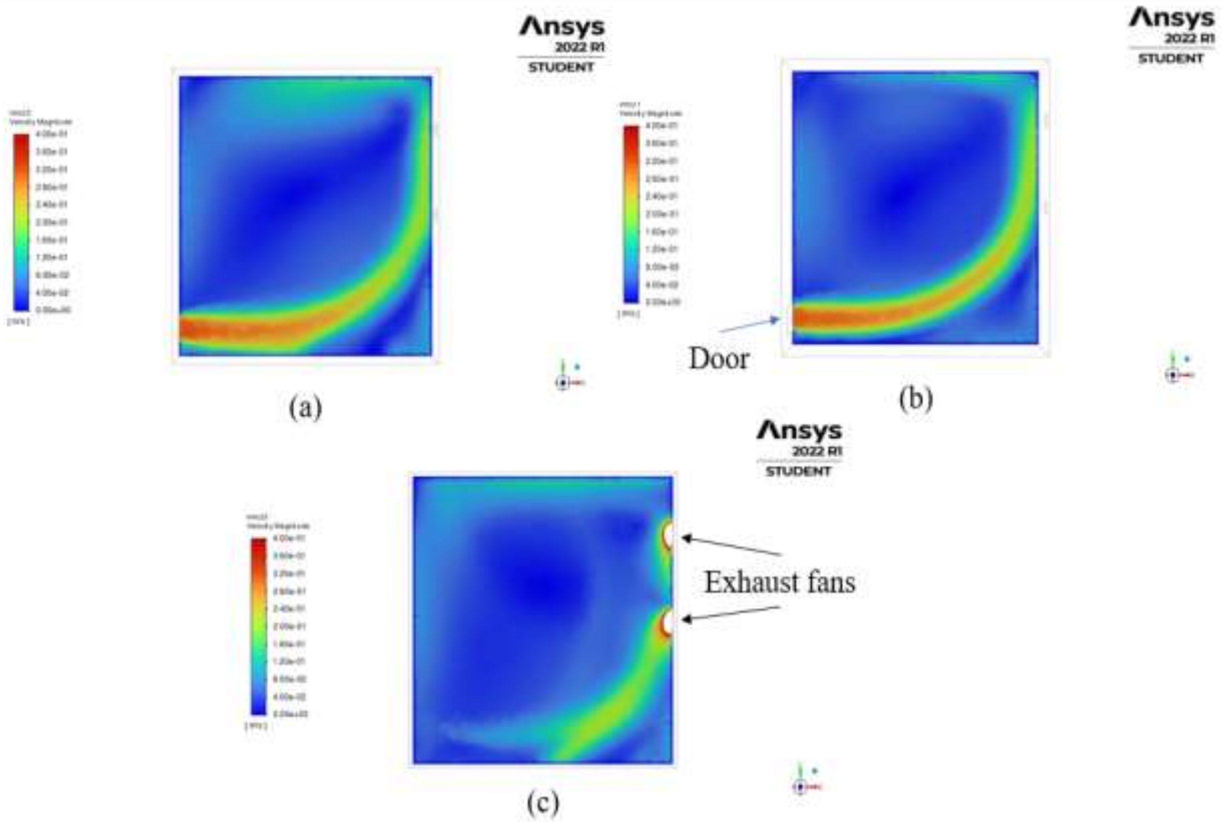


Figure 4 : (a),(b),(c)Velocity magnitude contours at a height of 1m (sitting height), 2m (standing height), 3m (exhaust fan height) along z in x-y plane.

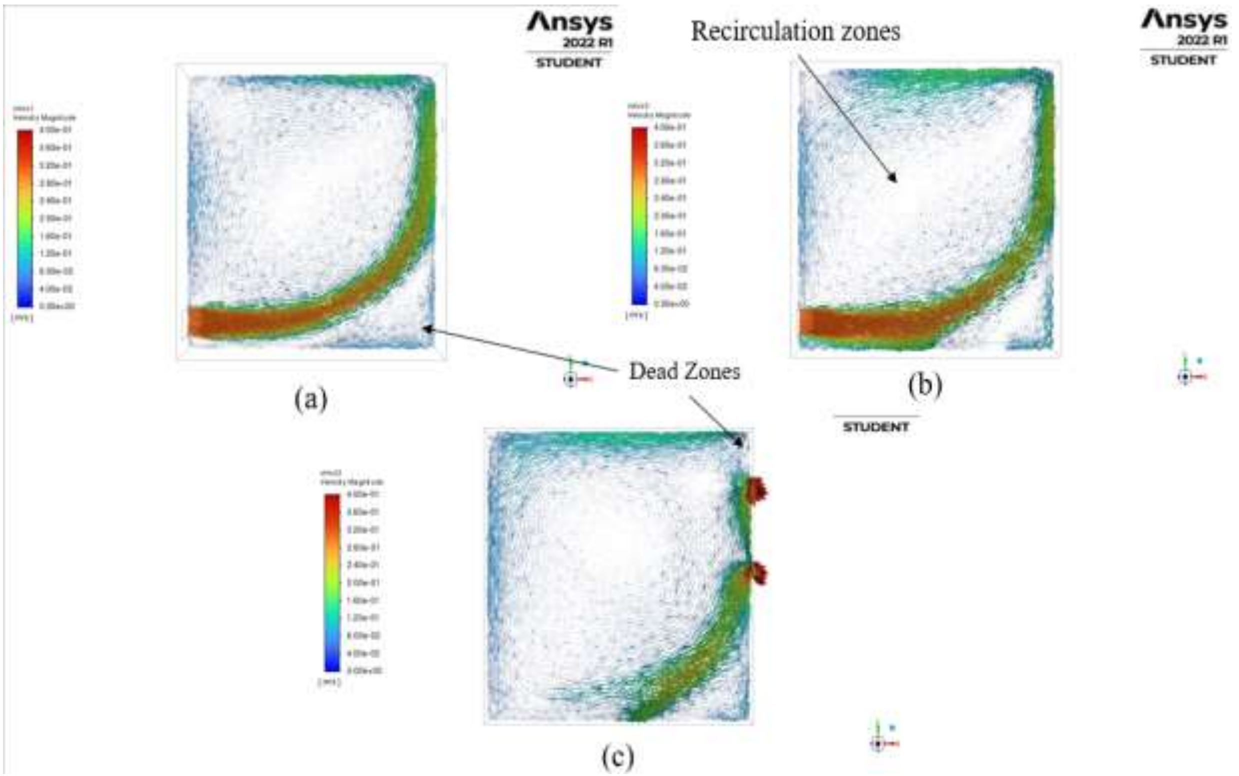


Figure 5 : (a),(b),(c)Velocity vectors at a height of 1m (sitting height), 2m (standing height), 3m (exhaust fan height) along z in x-y plane.

From the above results in figure8 and figure 9 it can be clearly visualised that how the air flow patterns is starting from door and moving towards the exhaust fans, a primary flow can be observed in the fluid domain. The recirculation zones visible in the vector plots, there is more chance of airborne infection at the centre of the classroom as the air residence will be more in this area. The age of air can be summarily proportional to the recirculation zones created in the low velocity areas.

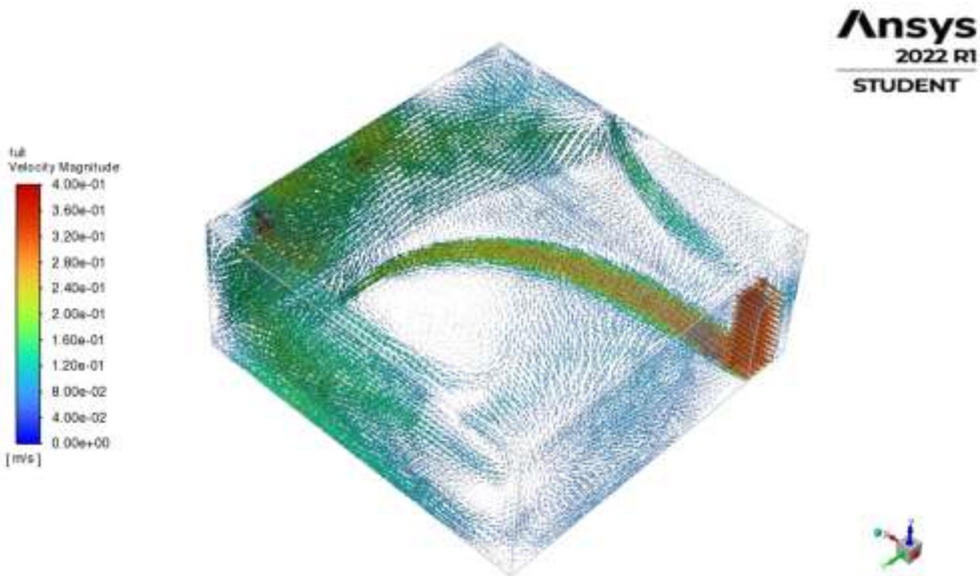


Figure 6: Velocity vectors in the fluid domain from doors to exhaust fans

From figure 10 the recirculation zones are clearly visible along z, on the walls and the corners of the room. The effect of exhaust fans can be visualised as the flow is moving towards the fans and then going out of the domain.

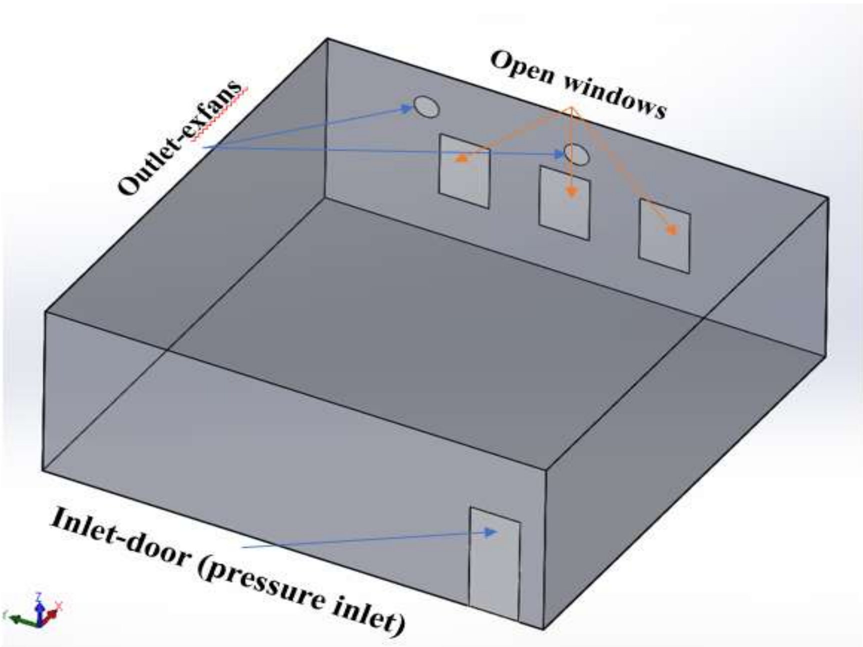
The velocity in the domain is ranging from m.0.4 to 0.6 m/s which is very less with compare to the ventilation and HVAC guideline. The velocity inside the domain can be increased with the introduction of fans which can become the study for further parts. To minimize these recirculation zones an optimum design and placement of exhaust fans played a major role in reducing the effect of recirculation and dead zones which results in optimum airflow with less residence time of air. The faster the air replaces in the room there is less chance of infection in the room. Through ventilation the objective can be achieved but it needs to paired with optimum inlet and outlet positions so that there is less chance of floe to recirculate and gets stagnate in a zone. The size of exhaust fans also affects the flow patterns so optimum design consideration should be selected for the fan size.

To compare it with a real scenario of a classroom the flow might differ as per the turbulence model selected in CFD. Many more tests are to run for similar case by selecting other turbulence models to know the relevance of flow field is not affected by the choice of turbulence models selected.

3.2 Case 2: Both Exhaust Fans working with open windows

3.2.1 Geometry Creation:

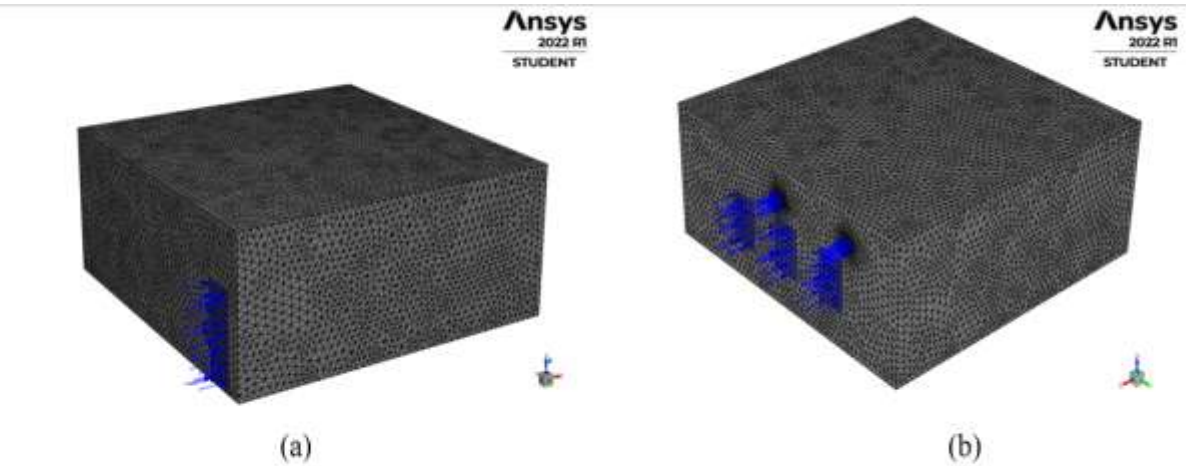
The geometry of this case can be referred above in figure 4. The room, door and the exhaust fans are kept with same dimensions as in case 1. The three windows are added in this case of size 1 x 1.5m and spacing between them is 1m. for the sake of simplicity the windows are considered as faces on the exhaust fans walls.



6. Figure 7: Geometry with windows

3.2.2 Meshing:

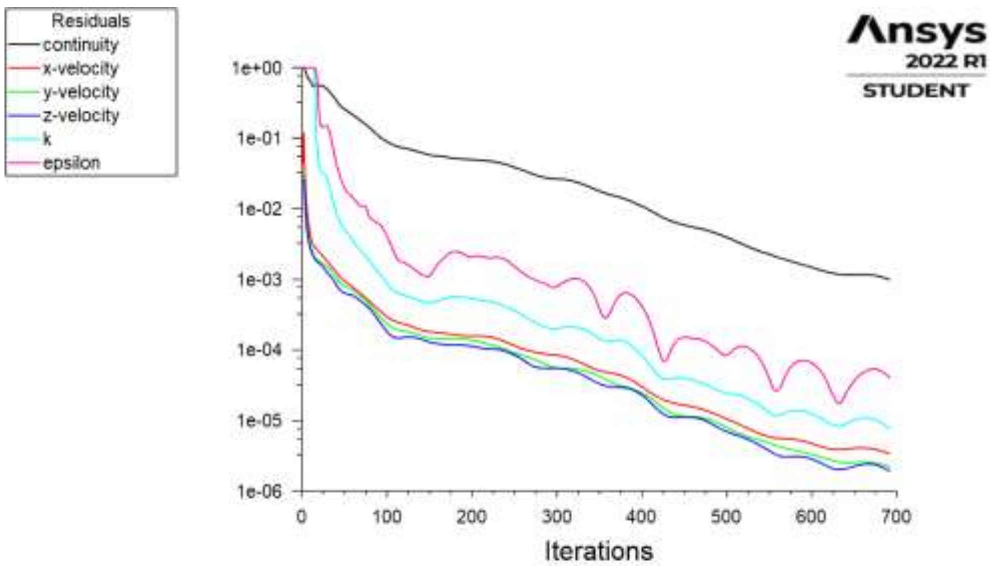
To visualise the airflow inside the defined geometry, tetrahedral mesh of 171k elements is used for the simulation. For discretization of this fluid domain the element size chosen is same of case 1 i.e., 0.2m. The mesh size is quite refined and as the mesh independent study depicts that the solution n does not change much after 171k cells.



7. Figure 8: Meshing for classroom with windows

3.2.3 Results:

The same turbulent kinetic energy(k)- epsilon model was considered with boundary conditions inlet-doors and inlet-windows at constant pressure inlet and exhaust fans as velocity outlet. The residuals were set below 10^{-3} for continuity and 10^{-4} for other parameters such as velocity, turbulent kinetic energy and dissipation rate.



8. Figure 9: Residual for the case 1

The solution got converged in less than 700 iterations. The residuals were also smooth and the downward behaviour of residuals depicts that the solution id robust and has very less effect on discretization errors. Below in figure 10 and 11 the velocity magnitude contours and vectors are visualized respectively. Through these results it can be visualised that the movement of airflow is similar in the case 2 as well as the primary flow generated starts from the door and move towards the exhaust fans. But in case 2 the amount of recirculation zones has increased due to the decrease in the velocity over the domain. Windows as being an inlet source to the domain creates an turbulent effect in that region and the air starts recirculating from the source to the exhaust fans. The pulling nature of exhaust fans created a spiral movement from the windows to the door which results in less velocity generated all over the domain specially the velocity from the door is reduced as the effect of recirculation region along the wall has been reduced. The velocity ranges from 0.04 to 0.2 m/s over the domain which is very less compare to the case1. The recirculation zone in this case has been shifted to more towards the end of the classroom which can be visualised from figure 11 below. More prominent dead zones are created as the overall velocity has decreased in the corners. As the upper movement is higher in the case where windows are open the destratification of air increases.

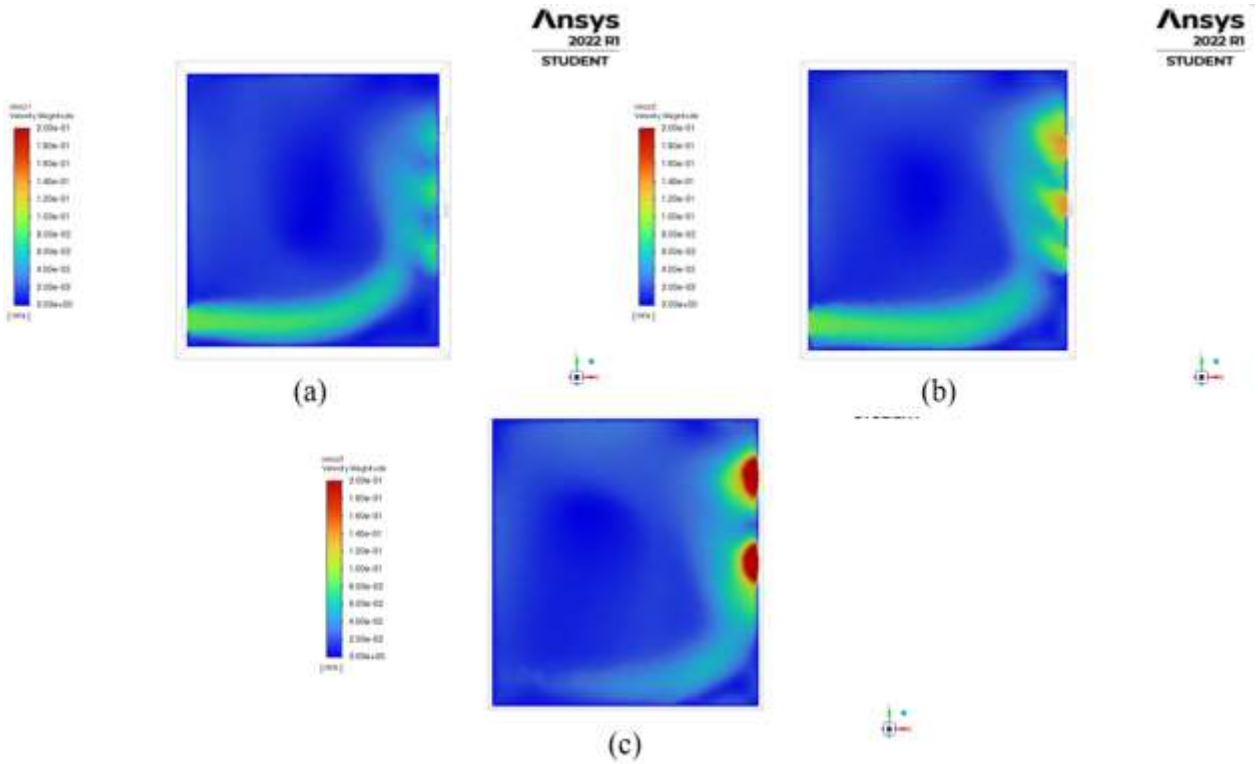


Figure 10 : (a),(b),(c) Velocity magnitude contours at a height of 1m (sitting height), 2m (standing height), 3m (exhaust fan height) along z in x-y plane.

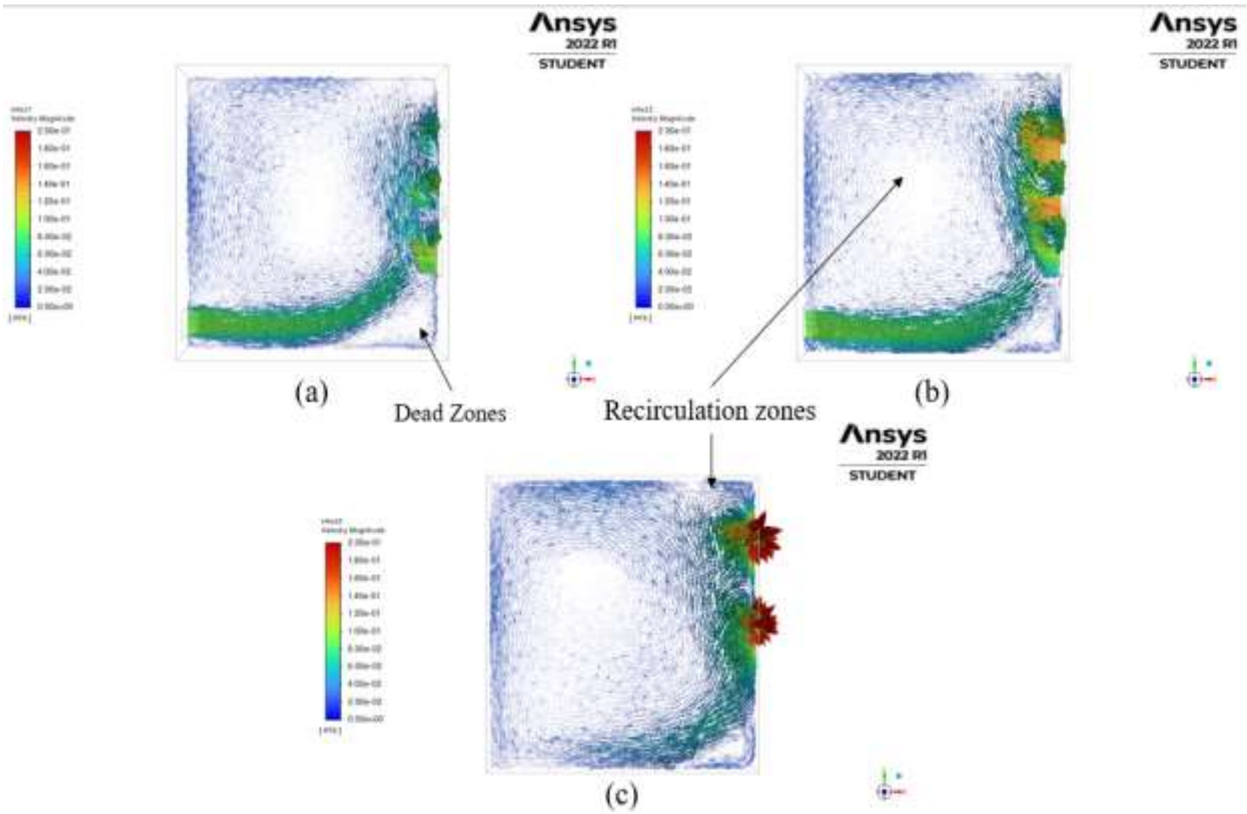
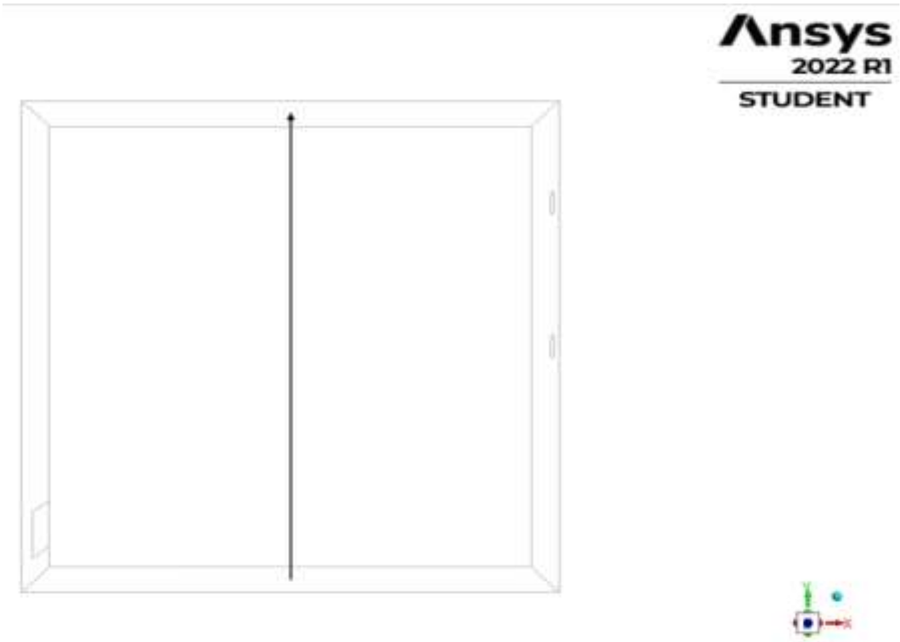


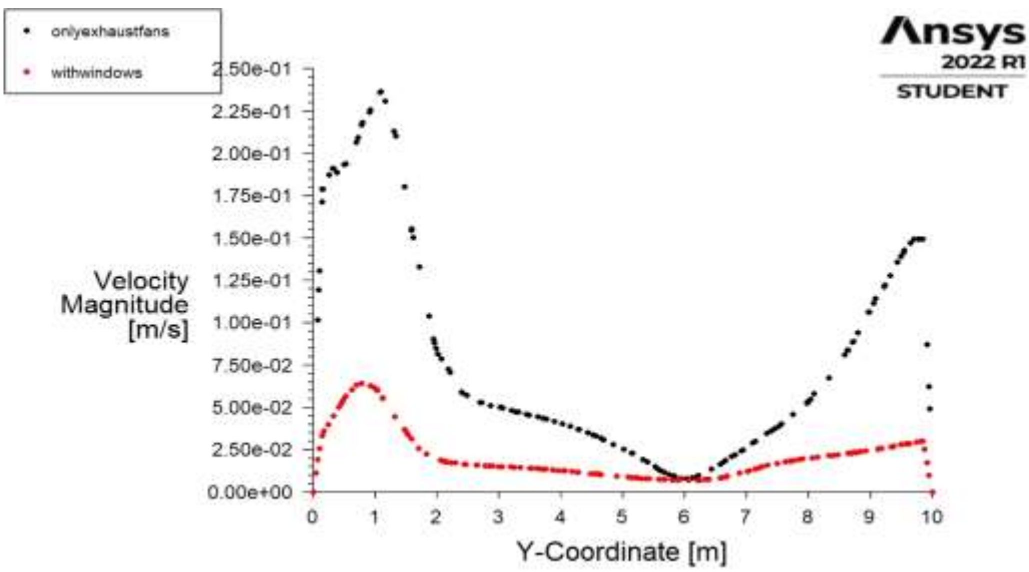
Figure 11 : (a),(b),(c)Velocity vectors at a height of 1m (sitting height), 2m (standing height), 3m (exhaust fan height) along z in x-y plane.

4 Comparison of domain with windows and without windows

From both the cases mentioned above it is clear that placement and design of optimum exhaust fans will result in proper ventilation in the domain as well as the air residing in the domain will be replaced more efficiently. In case of windows open the velocity in the domain decreased due to the recirculation zones created by the spiral effect formed due to the placement of exhaust fans just above the windows. The exhaust fans alone in case 1 provides better airflow pattern and visualisation with less recirculation and dead zones.



9. Figure 12: line drawn in the centre of the room



1. Figure 13: velocity magnitude and y coordinate plot over line in the centre of the room

To quantitatively define the comparison of two cases a virtual line is drawn at the centre of the room and velocity is plotted along that line as mentioned above in figure 12 and 13. The results shows that the decrease in the velocity near door with respect to case1. The velocity at door is 0.25 m/s in the case 1 where windows were closed and 0.05 m/s in case 2 where windows were kept opened. As the velocity decreases the age of air in the domain increase which results in increase chance of infection or prolonged inhalation of infective pathogens. Form the curve it clearly depicts the flow field around the line. At the centre of the classroom the velocity is lowest and it is highest at he door and back side of the classroom

5 Conclusion

In this work, flow induced by the operation of exhaust fan for a room has been studied. It can be concluded that the airflow generated by the pulling and pushing effect of exhaust fans is not same in both the cases where case 1 is without opening the windows and case 2 is with opening the windows. Interestingly the effect of opening the window has reduced the velocity in the domain conversely it should be the opposite. As the inlet numbers increased the more natural ventilation air should come in and there will be more a chance of replacing the existing air faster, but dure to the placements of exhaust fan just above the window ta spiral movement between the exhaust fan and window is created due to which the amount of air entering through window gets swirled out through exhaust fans. The major shortcomings of the exhaust fans are placement of fans as well as size. By reallocating it the flow of air can improved.

6. Future Scope

The effect of fans in the domain can also be part of future study as the mixing ventilation may increase the diffusivity via mixing the air homogeneously around the domain. The mixing of air also results in proper destratification of air which results in thermal comfort. If the virus is being transported through the fan then the study of size of exhaust fans vs the rate of air changes can be calculated.

7. References

(2022), A. (2022). *Ansys Fluent—CFD software | ANSYS 2022 R2.* http://www. ansys. com/ produ cts/ fluids/ ansys- fluent.

Aaron Foster, M. K. (2021). SARS-CoV-2 transmission in classroom. *Phys. Fluids*.

Abuhegazy M, T. K. (2020). Numerical investigation of aerosol transport in a classroom with relevance to COVID-19. *Phys Fluids 32:103311*(https:// doi. org/ 10. 1063/5.).

Anderson, J. D. (2012). In *computational fluid dynamics.* MCGRAW-HILL.

Ashkan Davani, H. A. (2022). Minimizing the COVID-19 spread in hospitals through optimization of ventilation systems. *Physics of Fluids*.

Babich, F., Cook, M., Loveday, D., Rawal, R., & Shukla, Y. (2017). Transient three-dimensional CFD modelling of ceiling fans. *Building and Environment, 123*, 37-49. doi:https://doi.org/10.1016/j.buildenv.2017.06.039

Bassiouny, R., & Korah, N. S. (2011). Studying the features of air flow induced by a room ceiling-fan. *Energy and Buildings, 43*, 1913-1918. doi:https://doi.org/10.1016/j.enbuild.2011.03.034

canada, S. Y. (2021). “Experimental investigation of indoor aerosol dispersion and accumulation in the context of COVID 19Effects of masks and ventilation”. *Physics of fluids*.

energy, E. e. (2001). *Cooling Your Home with fans and ventilation.* DOE/GO-102001-1278.

Gharbi, N. E., Absi, R., Benzaoui, A., & Bennacer, R. (2011). An improved near-wall treatment for turbulent channel flows. *International Journal of Computational Fluid Dynamics, 25*, 41-46. doi:10.1080/10618562.2011.554832

J, Y. W. (2001). *Handbook of flow visualization.* CRC Press.

Ju, L., & Du, Q. (2009). A finite volume method on general surfaces and its error estimates. *Journal of Mathematical Analysis and Applications, 352*, 645-668. doi:https://doi.org/10.1016/j.jmaa.2008.11.022

Li, Y., Yan, C., Yu, J., & Liu, H. (2017). A new high-accuracy scheme for compressible turbulent flows. *International Journal of Computational Fluid Dynamics, 31*, 362-378. doi:10.1080/10618562.2017.1365844

M.R.R.S. van Beest, F. A. (2022). Influence of indoor airflow on particle spread of a single breath and cough in enclosures: Does opening a window really ‘help’? *Atmospheric Pollution Research*.

Mikko Auvinen, J. K. (2022). High-resolution large-eddy simulation of indoor turbulence and its effect on airborne transmission of respiratory pathogens—Model validation and infection probability analysis. *Physics of Fluids*.

Naiping Gao, J. N. (2006). Transient CFD simulation of the respiration process and inter-person exposure assessment. *Building and Environment*.

Naiping Gao1, L. M. (2008). Distribution of Respiratory Droplets in Enclosed Environments under Different Air Distribution Methods. *Build Simul*.

Piomelli, U. (2010). Large eddy simulations. *International Journal of Computational Fluid Dynamics, 24*, 391-391. doi:10.1080/10618562.2010.546067

Pushpesh Singh, D. G. (2020). DESIGN AND CFD ANALYSIS OF CEILING FAN FOR REGULAR ROOM SIZE. *International Journal of Mechanical Engineering and Technology (IJMET)*.

*e*ISSN1303-5150 www.neuroquantology.com

Pushpesh Singh, G. V. (2020). DESIGN AND ANALYSIS OF HIGH VOLUME LOW SPEED FAN (HVLS) FOR ROOM SIZE 24M X24M X 18M. *International Journal of Mechanical and Production*.

Ramadan Bassiouny, N. S. (2011). Studying the features of air flow induced by a room ceiling-fan. *Energy and Buildings*.

Sasu Karttunen, M. K. (2020). Large-eddy simulation of the optimal street-tree layout for pedestrian-level aerosol particle concentrations – A case study from a city-boulevard. *ATMOSPHERIC ENVIRONMENT*.

Shuo Liua, A. L. (2018). Detailed experimental investigation of air speed field induced by ceiling fans. *Building and Environment*.

Son H. Ho, L. R. (2009). Thermal comfort enhancement by using a ceiling fan. *Applied Thermal Engineering,elsevier*.

Versteeg, H. (n.d.). AN introduction to CFD.

Wang H, L. M. (2014). Performance evaluation of air distribution systems in three different China railway high-speed train. *Build Simul*(https://doi. org/ 10. 1007/ s12273- 014- 0168-5).

Wenhua Chenb, S. L. (2018). Experimental and numerical investigations of indoor air movement. *Building and Environment,elsevier*.

wiki. (n.d.). Retrieved from wikipedia: https://en.wikipedia.org/wiki/Computational_fluid_dynamics

Yin Y, G. J. (2011). Distributions of respiratory contaminants from a patient with different postures and exhaling modes in a single-bed inpatient room. *Build Environ*(https:// doi. org/ 10. 1016/j. build env. 2010. 07. 003).

Yoshihisa Momoi, K. S. (2004). Modeling of Ceiling Fan Based on Velocity Measurement for CFD Simulation of. *Journal of Environmental Engineering*.