**Proceedings of the 8th International and 47th National Conference on Fluid Mechanics and Fluid Power (FMFP)**
**December 9-11, 2020, IIT Guwahati, Assam, India**

**FMFP2020-168**

# Numerical study on various techniques of obtaining negative pressure room using OpenFOAM

**Mr. Divyesh Variya[1], Dr. Janani Srree Murallidharan[2]**

[1]FOSSEE, Indian Institute of Technology Bombay, Mumbai 400076, India
[2]Department of Mechanical Engineering, Indian Institute of Technology Bombay, Mumbai 400076, India

## ABSTRACT

A negative pressure room is used to treat patients, affected by airborne ailments, like H1N1 Swine Flu, Ebola, SARS, and COVID-19. AIIRs (Airborne infection isolation Rooms) prevent secondary cross-contamination inside hospitals. This paper aims to illustrate various arrangements to obtain negative pressure inside a health-care facility. Using computational fluid dynamics, pressure, and airflow patterns can be predicted to set the position of the air inlet and outlet. With the help of OpenFOAM CFD software, a transient simulation carried out for various possibilities of maintaining negative pressure inside a room effectively.

**Keywords:** Negative Pressure, AIIRs, CFD, Airborne disease, OpenFOAM

## I. INTRODUCTION

Over the past few decades, airborne diseases such as Tuberculosis, severe acute respiratory syndrome, Covid-19 and Ebola has harmed countless lives. Certain types of diseases are controlled by treating patients in a negative pressure room, which is also called a quarantine room or AII Room. In the negative pressure room, engineers suggest maintaining at least -2.5 Pascal of negative gauge pressure. In some countries, it is not possible to build an entirely new structure of health-care facilities in big continents like India. In such regions, it is always a challenge to contain/control such type of airborne diseases. In such a case, the role of computational fluid dynamics becomes very crucial to develop a new method to convert an existing health care facility to an isolation health care facility temporarily.

## II. LITERATURE REVIEW AND OBJECTIVE

Prasad Mahajan et al. [3] analyzed the steady-state conditions of a negative pressure room. They studied a multiphase simulation with air flowing from the inlet, which is at the leg side, and going out of the outlet, which is near to the patient's head. By plotting contours of temperature, velocity, pressure, and $CO_2$ concentration, various conclusions are made. The study predicted that the $CO_2$ exhaled by the patient does not spread inside a negative pressure room. The simulation performed using Ansys Fluent, which uses the finite volume method for the numerical study.

Shih Y.C. et al. [7] studied the dynamic airflow simulation inside an isolation room. They analyzed the effects of a moving person on the air distribution inside the room. Based on their observations of the velocity, pressure, and contaminant fields, they arrived at two conclusions (1) The air distribution is easily affected by the moving person. However, the airflow returns to the original state quickly. Thus, the contaminants near the patient are not affected by the moving speed. (2) The opening and closing of a door have an earnesta significant effect on internal pressure and velocity distributions. It causes sudden rises and drops of the internal pressure during the periods of opening and closing the door. CFD simulation involved the use of dynamic meshing and transient setup. $\kappa - \epsilon$ turbulence model and $CO_2$ as a contaminant source adopted in the simulation. The simulation is performed using Ansys Fluent numerical model, which uses the finite volume method.

Chow et al. [6] investigated the ventilation system of a hospital operating theater. Specifically, they have investigated the effect of creating a temporary room near to the patient's bed (surgical field). They Found that air-distribution systems provide an optimum effect within the surgical area rather than in the entire room. The added advantage is that maintaining negative pressure within the surgical area reduces power consumption.

Cheong K.W.D et al. [2] analyzed the airflow and pollutant distribution patterns in a "negative pressure" isolation room using CFD modeling based on three ventilation strategies. Strategy 1 has two air supply diffusers and two extract grilles mounted on the ceiling. Strategy 2 retains the air supply diffusers in Strategy 1 but relocates the two extract grilles to the wall behind the bed at 0.3 m above the floor level. Strategy 3 has the same layout as Strategy 2, except the ceiling diffusers are replaced by supply grilles and relocated closer to the wall behind the bed. In all the three ventilation strategies, the locations of supply diffusers and extract grilles were changed; and numerical simulations performed. The study found that ventilation strategy 3 is best with pollution removal efficiency values exceeding 1; and it has the lowest exposure level of the three locations.

Guillermo Giraldo [1] investigated different outlet positions to reduce the risk of bacteria being spread. The case study also considered thermal comfort and fresh air velocity conditions inside the hospital room. The numerical analysis shows that the best position of the outlet is near the patient. Also, in some cases, recirculation occurs inside the room. Recirculation of air is acceptable only if contaminated air is not involved in the recirculation zone.

Airborne Infectious Disease Management Methods for

Temporary Negative Pressure Isolation [5] gives various practical techniques to convert a present health care facility into a short-term quarantine facility. The study only covers practical applications without any numerical simulations.

## III. MATERIALS AND METHODS

The numerical simulations were carried out to maintain negative pressure inside a health care room. An open-source CFD software, OpenFOAM is used to discretize the geometry and solve the Naiver Stokes equations. Initially, a room with dimensions given in table 1 created using blockMesh utility; and geometry of Bed and Patient snapped from the room, snappyHexMesh utility used. Both utilities used for meshing are of OpenFOAM software, which allows discretization of the domain in hexahedral blocks. Pre-written FVM based buoyantPimpleFoam solver is used to simulate the problems.

Here, three cases are studied. The first case is a validation case, which is set up by Prasad Mahajan [3]. The same case simulated using OpenFOAM in this paper. In addition to the validation case, two different cases with an anteroom are studied.

### A. Solver and Equations

To simulate the problem, a buoyancy and Boussinesq term based solver, which can also handle the $\kappa - \epsilon$ turbulence model used. To get more accurate results, Boussinesq Approximation and energy equations solved in Navier stokes equations.

*1) **Navier Stokes Equation**:* The navier stokes equations are set of mass, momentum and energy equations.

**Continuity equation:**
The general simplified form of continuity equation states that,

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

**Momentum equation:**
The momentum equation,

$$\rho \nabla \cdot (uu) \quad = \quad -\nabla p + \nabla \cdot (\mu \nabla u) + \rho g \tag{2}$$

**Energy equation:**
For the constant material properties, Energy equation can be written as,

$$\nabla \cdot (Tu) \quad = \quad k_{eff} \nabla^2 T \tag{3}$$

Where,

$$k_{eff} = k + k_t = \frac{\nu_0}{pr} + \frac{\nu_t}{pr_t}$$

$$pr = \frac{c_p \mu_0}{k} \qquad pr_t = \frac{c_p \mu_t}{k_t}$$

**Boussinesq Approximation:**
In the momentum equation, a buoyancy term $\rho g$ indicates variation in density
The relation between density and temperature can be written as,

$$\Delta \rho \quad = \quad -\rho_0 \beta (T - T_{ref}) \tag{4}$$

Where, $\beta$ is coefficient of thermal expansion
$T_{ref}$ is reference Temperature
So, the buoyancy term can be rewritten as,

$$\rho g \approx [1 - \beta (T - T_{ref})] g \tag{5}$$

*2) **PIMPLE Algorithm**:* The PIMPLE algorithm is a hybrid of SIMPLE and PISO loops. It is more suitable for unsteady simulations. The PIMPLE algorithm allows to run a simulation with large time steps with multiple pressure correctors in the loop.

*3) $\kappa - \epsilon$ **Turbulence model and wall function**:* Widely known $\kappa - \epsilon$ turbulence model for the HVAC system is used to incorporate turbulence and wall functions. Its lower computational cost make it more reliable for HVAC systems.[4]

The $\kappa - \epsilon$ turbulence model solves two additional equations, for turbulent kinetic energy $\kappa$ and rate of dissipation of turbulence energy $\epsilon$.

$\kappa$ can be initialize by,

$$\kappa \quad = \quad 1.5 (U_\infty I)^2 \tag{6}$$

$$I \quad = \quad 0.16 \cdot Re^{-1/8} \tag{7}$$

Where, $U_\infty$ is velocity
$I$ is turbulent intensity

$\epsilon$ can be initialize by,

$$\epsilon \quad = \quad \frac{{C_\mu}^{3/4} \kappa^{1.5}}{0.07 \cdot L} \tag{8}$$

Where, $C_\mu$ is an empirical constant. Its value is 0.09.

It is challenging to keep $y^+$ in the viscous sub-layer. This approach leads to a requirement of high cell numbers, which means high computational resources are needed. So, Mesh is kept in a log law region, which is $30 < y^+ < 300$. Here, the wall function approach is used to ensure the accuracy of the results.

### B. Cases

Three different cases tested in this study. First, the best suitable position for the inlet and outlet identified using a simple room geometry. The room is three dimensional with a patient and a bed inside it. A fixed inlet & outlet is shown in figure 1. Simplified geometry of patient and bed are taken into account. The dimensions of the Room, Bed, and Patient are specified in the table 1. Boundary conditions are available in table 2.

**Table** 1: **General Dimensions**

| | Units | Width | Length | Height |
|---|---|---|---|---|
| Room | $m$ | 4 | 4 | 2.6 |
| Patient | $m$ | 0.6 | 0.1 | 1.7 |
| Bed | $m$ | 1 | 1.75 | 0.6 |
| inlet | $m$ | 0.8 | – | 0.2 |
| Outlet | $m$ | 0.8 | – | 0.2 |

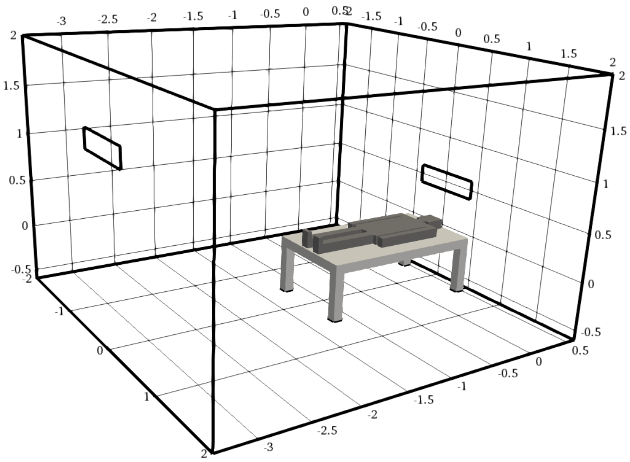

**Figure** 1: **Base set-up of AII-Room.**

**Table** 2: **Boundary Conditions and Parameters**

| Boundary | Units | Value |
|---|---|---|
| Air Temperature | $K$ | 295.15 |
| Inlet Velocity | $m/s$ | 0.1 |
| Outlet Pressure | $pa$ | $-8$ |
| Turbulence intensity | % | 15 |

Giraldo [1] mentioned the comfort velocity value for a patient. To achieve comfort airflow inside a health-care room, inlet air should be restricted and outlet pressure should be such that inside room pressure becomes approx -3 pascal. The higher outlet pressure doesn't give expected pressure inside the room, and lower outlet pressure causes additional velocity circulation, which may lead to discomfort for the patient. Hence, the best suitable inlet-outlet boundary conditions given for the room stated in table 2. The focus of this study is to compare the three different designs and hence a single optimum velocity value based on the optimum pressure value is deemd sufficient for the scope of this paper. Further detailed parametric studies will be undertaken later.

Note: Here, it is assumed that there is no leakage of air outside the domain. So, the room used here is fully airtight, and no other air is going in or out other than from the inlet and outlet specified in the case. The pressure value specified is gauge pressure, not the absolute pressure.

*1) **Case 1: A validation case with 30$^o$ air inlet**:* A 30$^o$ inlet is specified. The problem setup and methodology are shown in figure 2. The same case used by Prasad Mahajan [3] validated here using OpenFOAM.

A grid independence study was performed using two different cell sizes. One case ran with 14,76,122 cells and another with 37,63,550 cells. The simulation results (velocity and pressure) in both cases were found identical with approx. 0.5 % of error. The residuals convergence criteria kept to $10^{-4}$ and achieved in both cases. Hence, a case set up with a lower number of cells is taken into account to reduce computational cost.

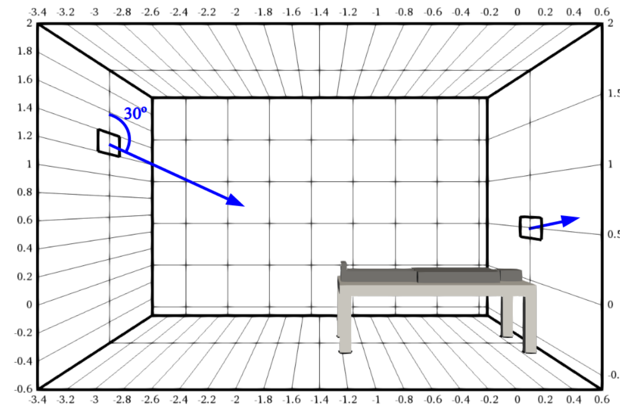


**Figure** 2: **Validation case set-up.**

*2) **Case 2: Ante-room to convert existing room to AIIR**:* Instead of building a complete new health-care facility with a negative pressure room, it is always better if we can convert the existing health-care facility to an AII Room. To convert an existing health-care facility into a temporary negative pressure room, an ante-room is attached. The concept of anteroom is to separate two domains with different inside conditions. In this case, temporary walls created using plastic or cotton sheets. Inlet and outlet only specified to the anteroom. The only air going inside the room should be from the leakage of the door. The schematic diagram of the case is shown in figure 3.

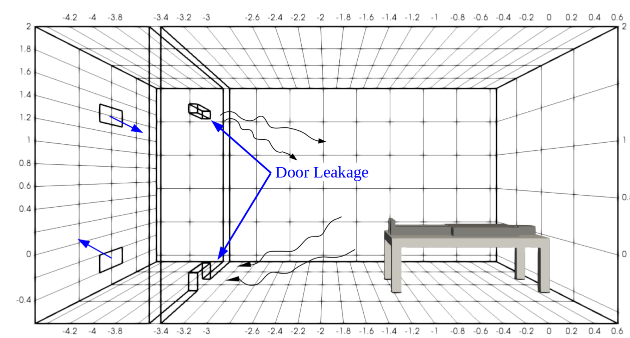


**Figure** 3: **Anteroom set-up with door leakage.**

The specifications of the anteroom and inlet-outlet positions are prime factors in this case. The geometry, size, and location of the anteroom, inlet, and outlet will affect the pressure inside the main room. The specifications used in this case are shown in table 3.

**Table** 3: **General Dimensions**

| Parameter | Units | Length | Width | Height |
|---|---|---|---|---|
| Anteroom size | $m$ | 4 | 1 | 2.6 |
| Leakage size at floor level | $m$ | – | 0.8 | 0.2 |
| Leakage size at top of door | $m$ | – | 0.8 | 0.1 |

*3) **Case 3: Ante-room with cyclic pair**:* Using the benefits of both cases 1 & 2 an additional cyclic paired case is created. An inlet and outlet create negative pressure in the room, and a cyclic pair makes sure that contaminated air quickly sucked out from the cyclic patch. As shown in figure 4, one end of the cyclic pair is near to the patient and another is near to the outlet where the vacuum pump is attached.

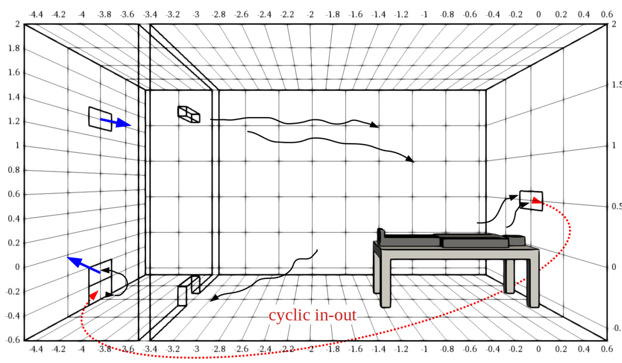


**Figure** 4: **Anteroom with cyclic pair.**

## IV. RESULTS AND DISCUSSION

For all three cases, the velocity vector and pressure contours plotted. There is not much notable change in temperature in this study. Hence, temperature contours are not stated in the results. However, it is required to incorporate the Boussinesq approximation and energy equation to get more accurate pressure and velocity profiles.

### A. Negative pressure contour comparison

Using negative pressure of -8 pascals at an outlet, the constructive negative pressure obtained in the room is -3.5 pascals. Case 1 succeeds in achieving negative pressure by just restricting airflow in the domain and forcing outflow using a vacuum pump. This phenomenon enables the desired negative pressure to reach within a reasonable timescale (∼20 sec).


**Figure** 5: **Pressure contour in case 1.**

Pressure contour found in Prasad Mahajan's [3] study is approx -3.2 pascals. Case 1 results observed approx 9 % variation in pressure due to different meshing and simulation techniques used by the software.

Case 2, which has the anteroom attached, gives the best effective pressure distribution over the room. The negative pressure in the domain is largely -3.6 pascals, which are more uniform than in Case 1. The lesser variation of negative pressure in the room gives higher comfort to the patient. The power consumption here is relatively lesser than other cases due to lesser air movement inside the room. Additionally, the desired negative pressure obtained in a shorter time frame in comparison to Case 1.

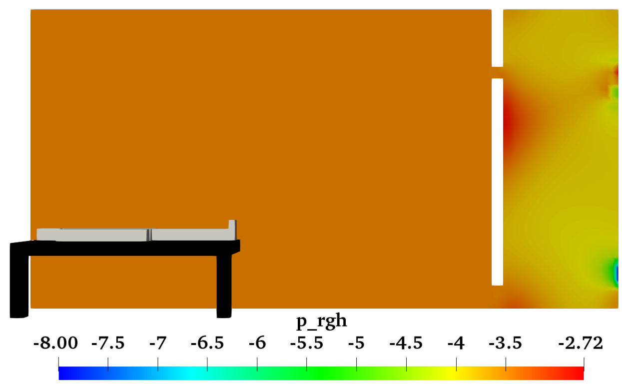


**Figure** 6: **Pressure contour in case 2.**

The hybrid model of cases 1 & 2 gives the best negative pressure distribution in the main room. That is substantiated in the following discussion.

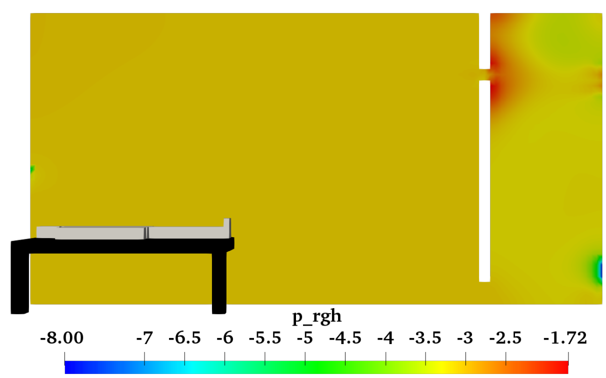


**Figure** 7: **Pressure contour in case 3.**

### B. Velocity vector comparison

Velocity vector contour shows that, in case 1, that fresh air is received by a patient in fair amounts. Constant air supply to the patient is visible. There appears to be a recirculation zone within the room as also observed by Giraldo [1]. The recirculation zone is likely because the incoming velocity is impingeing on the floor with some part of it deflecting way from the patient. However, we are not convinced that the recirculation strength which is being displayed is as significant as predicted. We are looking through literature and also the post-processing setting to gain clarity on this matter. However, the general physics and qualitative trends of velocity and pressure trends are accurate as only 9 % variation as detected compared to literature. In the subsequent cses, we have presente recirculation as well, but would like to only relatively compare it with case 1 and do so only for providing a qualitative understanding.

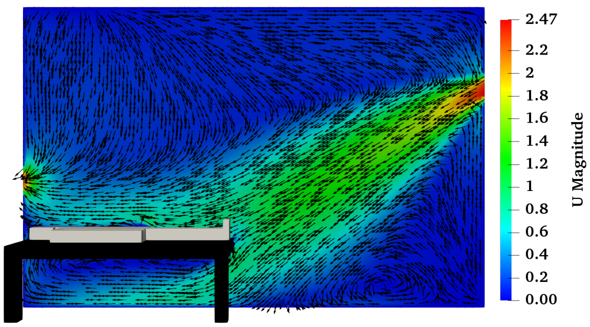


**Figure** 8: **Velocity contour in case 1.**

In case 2, where anteroom is attached, the velocity profile with vector shows that extreme less air circulation is obtained. This might prove uncomfortable for the patient

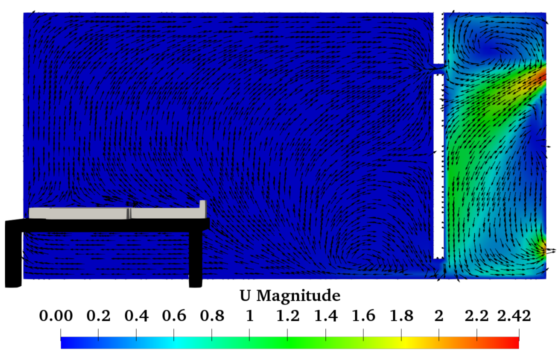


**Figure** 9: **Velocity contour in case 2.**

Case 3, has an additional pair of cyclic patches. This is implemented to set-up a constant air flow similar to case 1. This achieves reasinable airflow near the patient, and also achieves the required pressure drop

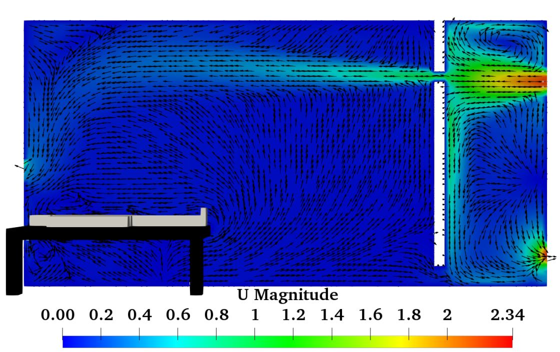


**Figure** 10: **Velocity contour in case 3.**

## V. CONCLUSIONS

The techniques to obtain negative pressure inside a room is satisfied in all three cases. With the same power vacuum pump, -3.5 pascal negative pressure maintained. In case 1, from the simulation, it is concluded that it takes 20 seconds to stabilize the flow in the domain.

In case 2, an anteroom joined to convert the existing health-care room into a negative pressure room. An ante-room is made of temporary plastic sheets, which will create a temporary structure, where an inlet, and outlet are attached. The benefit of this structure is that it does not just generate a negative pressure inside the main room but restricts airflow in it. That will save others from being infected by patients. The pressure plots show that the same negative pressure can be reached. This can be achieved with a minimum vacuum pump effort as suction is effected in the ante-room only.

In case 3, the advantage of cases 1 & 2 is taken into account by adding a cyclic boundary near the patient and outlet. It took the same time to stabilize the flow in the room as it took in case 1 but, -3.5 pascal pressure with the more stable distribution obtained. The risk of catching a disease is reduced due to two outlet zones. Any contaminated air exhaled by a patient will be sucked out from the cyclic patch near the bed. If any health worker exhales contaminated air, that is taken out from the floor level door leakage. More importantly, a stedy flow is set-up near the patient to aid is comfort.

Hence, all three strategies work very well to obtain negative pressure in the domain; but the minimum power consumption is in case 3. Therefore, case no. 3 is the best suitable setup.

## ACKNOWLEDGEMENTS

The authors would like to express their gratitude for support from FOSSEE, IIT Bombay and Govt. of India. The project is funded by NMEICT, Ministry of Education, India.
Authors also thank fellow members of our team, Mr. Ashley, Ms. Padmini, Ms. Swetha and Mr. Shouvik for their valuable inputs and suggestions.

## NOMENCLATURE

| | | |
|---|---|---|
| $\kappa$ | Turbulent kinetic energy | [$m^2/s^2$] |
| $\epsilon$ | Dissipation | [$m^2/s^3$] |
| $u$ | Velocity | [m/s] |
| $\rho$ | Density | [$kg/m^3$] |
| $p$ | Pressure | [Pa] |
| $\mu$ | Kinematic viscosity | [$m^2/s$] |
| $T$ | Temperature | [K] |
| $Pr$ | Prandtl no. | – |
| $k_{eff}$ | Effective thermal conductivity | [W/(m·K)] |
| $\nu$ | Dynamic viscosity | [$N \cdot s/m^2$] |
| $c_p$ | Specific heat capacity | [J/(kg·K)] |
| $\beta$ | Thermal expansion coefficient | [$K^{-1}$] |
| $U_\infty$ | Velocity | [m/s] |
| $I$ | Turbulent intensity | – |

## References


[1] Guillermo Giraldo, *How to use cfd to simulate airflow in an operating room*, https://www.simscale.com/blog/2018/12/cfd-airflow-operating-room/, (accessed: July 15, 2020).

[2] Cheong K.W.D. and Phua S.Y., *Development of ventilation design strategy for effective removal of pollutant in the isolation room of a hospital*, Building and Environment **41** (2006), 1161–1170.

[3] Prasad Mahajan, Arun Saco S, R. Dinesh Kumar, and Thundil Karuppa Raj R., *Airflow simulation of an isolation room using cfd technique*, International Journal of Pure and Applied Mathematics **118** (2018), 4261–4269.

[4] Pankaj Mishra and K R Aharwal, *A review on selection of turbulence model for cfd analysis of air flow within a cold storage*, IOP Conference Series: Materials Science and Engineering **402** (2018), 012145.

[5] Minnesota Department of health, *Emergency preparedness and response*, https://www.health.state.mn.us/communities/ep/surge/infectious/airbornenegative.pdf, (accessed: July 15, 2020).

[6] Chow TT, Ward S, Liu JP, and Chan FCK., *Airflow in hospital operating theatre — the hong kong experience*, Proceedings of healthy buildings 2000 design and operation of HVAC systems **2** (2000), 419–424.

[7] Shih Y.C., Chiu C.C., and Wang O., *Dynamic airflow simulation within an isolation room*, Building and Environment **42** (2007), 3194–3209.