FOSSEE, IIT Bombay
OpenFOAM Case Study Project
June 20, 2020

CFD study of ventilation in a room maintained under negative-pressure to prevent airborne contamination

Shouvik Ghorui

Jadavpur University, Kolkata

Abstract

Isolation rooms are used in hospitals for patients who suffer from diseases like tuberculosis, COVID-19, and severe acute respiratory syndrome. It is very crucial to protect doctors, nurses, and other health-care workers from patients with infectious diseases by the effective ventilation system. This case study studies different inlet-outlet configurations in order to determine the most suitable combination that will help prevent air-borne contamination in a negative pressure isolation room. Computational Fluid Dynamics is a technique that is used to analyse the indoor environment of a ventilated room and overall ventilation of air distribution systems. *blockMesh* utility is used for meshing a three-dimensional room of size 4*4*2.5 m^3 Bed and patient are modelled in CAD software. The final finished mesh is done using *snappyHexMesh* utility of OpenFoam. Discretized conservation equations like continuity, momentum, energy and turbulence are solved simultaneously using CFD Open Source package **OpenFOAM® V-7.0** with *buoyantPimpleFoam* solver to simulate the flow Physics.

Keywords:- CFD, OpenFOAM, Negative pressure ventilation room, COVID-19

1. Introduction

Over the last few decades, the consciousness for air flow inside the room is increasing and its influence on the design of the air-conditioning and mechanical ventilation (ACMV) system is also an important factor. This is especially crucial in hospitals where air-borne transmission of contaminated air is the second most prevalent cause of causing disease for patients, healthcare workers, and visitors. The role of the ACMV system in hospitals is more vital than just the provision of thermal comfort. In many cases, proper air-conditioning is a factor of paramount importance in patient therapy and in some cases, it is the major treatment. The quality of the overall hospital environment depends on control of certain factors like pressure, temperature, velocity, and indoor air quality. These variables are analyzed using the Computational Fluid Dynamics technique. The role of CFD is important in the prediction of ventilation performance in buildings. The isolation room should maintain negative air pressure with respect to the surroundings. The negative pressure is maintained by extracting a higher amount of air than that is supplied(Fig:1). This pressure differential causes air to flow into the room through various leakage areas (e.g. the perimeter of doors and windows, utility/fixture penetrations, cracks, etc.) instead of the air flowing outwards.

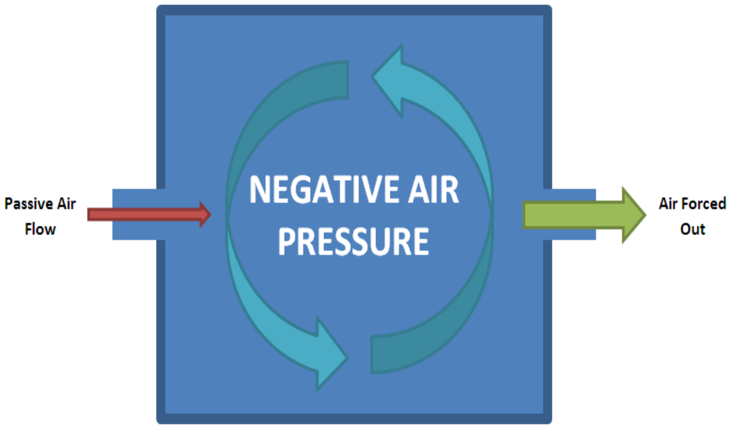


Fig: 1 physics for this case
Source: wikipedia

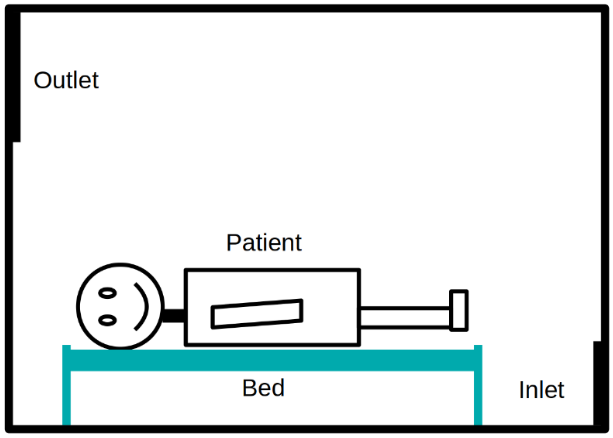


Fig: 2 rough geometry for this case

Literature review:

Prasad Mahajan et al. [1] Airflow Simulation of an Isolation room using CFD Technique. They analyzed the steady-state conditions of the room. By plotting contours of temperature, velocity, pressure, and CO_2 concentration various conclusions are Made. The simulation was performed using Ansys Fluent numerical model is based upon finite volume. ***Shih Y.C. et al.*** [2] studied the dynamic airflow simulation within an isolation room. They analyzed the effects of a moving person on the indoor air distribution including velocity, pressure, and contaminant fields. CFD simulation involved the use of dynamic meshing and transient setup. K-ε turbulence model and CO2 as a contaminant source was adopted in the simulation. The simulation was performed using Ansys Fluent numerical model is based upon finite volume. ***Chow et al.*** [3] investigated the ventilation system of a hospital operating theatre and found that the optimum supply air-distribution systems provide the desired effects within the surgical field rather than in the entire room. ***Cheong K.W.D et al.*** [4] “Development of ventilation design strategy for effective removal of pollutant in the isolation room of a hospital” in this paper they analyzed the airflow and pollutant distribution patterns in a “negative pressure” isolation room by means of CFD modelling based on three ventilation strategies. In all the three ventilation strategies the locations of supply diffusers and extract grilles were changed and numerical simulations performed on these strategies were compared. The aim was to detect which strategy is better for pollutant removal. It was found that Ventilation Strategy 3 is best with pollution removal efficiency values exceeding 1 and it has the lowest exposure level at the three locations. ***Guillermo Giraldo*** [7] investigated different outlet positions to reduce the risk of bacteria to be spread in an operation theater room. Case study also considered thermal comfort and fresh air velocity conditions inside the hospital room. In this case, we validate against [7], and moving further, we look at positions of inlet- outlet combination that will be most suitable.

2. Problem statement:

In this case, there is a room with an inlet and an outlet along with a patient and a bed. The air is flowing from the inlet to the outlet and to maintain negative pressure inside the room a higher proportion of the air is extracted than that is supplied. Patient is also releasing heat from the body. Thermal comfort near the patient has to be ensured and also pressure inside the room has to be ensured to remain negative always. The main analysis of this case is to find the best suitable position of inlet and outlet and along with this suitable bed position such that it reduces the chances of the infection to other people (doctor, nurse, patient visitors). For this case The flow conditions are shown as in Table 1:

Fluid property	Value
Kinetic viscosity, fluid(μ), Pa.s	$1.825 * 10^{-5}$
Reference Density of the fluid(ρ), kg/m^3	1
coefficient of thermal expansion, K^{-1}	0.0034
Prandtl number	0.7039

Table 1: Details of fluids property

3. Mathematical modelling:

The initial temperature of the fluid is taken as 295.15K and all the thermophysical properties are taken for 295.15K, the density of air is considered as a variable because it changes by changing the temperature. So to incorporate the variable density, Boussinesq approximation is used, according to this approximation change in density is assumed as a negative linear function of temperature. The approximation is accurate when density variations are small as this reduces the nonlinearity of the problem.

Mathematically,

$$\nabla\rho = -(\rho_0 \beta \Delta T)$$

Where

- ρ_0 is the initial density of the fluid
- β is the coefficient of thermal expansion
- $\Delta T = T - T_{reff}$
- The above approximation mentioned in the equation is valid only when $\Delta\rho \ll \rho_0$

3.1 Continuity equation:

$$\frac{1}{\rho}\frac{D\rho}{Dt} + \nabla.u = 0$$

3.2 Momentum equation:

In the The Boussinesq Approximation, It assumes that variations in density have no effect on the flow field, except that they give rise to buoyancy forces. In more practical terms, this approximation is typically used to model liquids around room temperature, natural ventilation in buildings, or dense gas dispersion in industrial set-ups.

N-S equation for general compressible flow case is :

$$\rho(\frac{\partial \rho}{\partial t} + u.\nabla u) =- \nabla p + \nabla.\{\mu(\nabla u + (\nabla u)^T) - \frac{2}{3}\mu(\nabla u)I\} + \rho g$$

where u is the fluid velocity, pis the fluid pressure, ρ is the fluid density, μ is the fluid dynamic viscosity, I is the identity matrix, and g is the acceleration due to gravity. The Navier-Stokes equations are solved together with the continuity equation. The Boussinesq approximation states that the density variation is only important in the buoyancy term,and the rest of the term it can treat as constant.

So now NS equation will look like

$$\rho_0(\frac{\partial \rho}{\partial t} + u.\nabla u) =- \nabla p + \nabla.\{\mu(\nabla u + (\nabla u)^T) - \frac{2}{3}\mu(\nabla u)I\} + \rho g$$

where the temperature and pressure-dependent density, ρ, have been replaced by a constant density, ρ_0 , except in the body force term representing the buoyancy force. From the Boussinesq approximation continuity equation can be rewrite as the incompressible form

$$\nabla.u = 0$$

because the magnitude of $\frac{1}{\rho}\frac{D\rho}{Dt}$ is small with respect to the velocity gradients $\nabla.u$
And by neglecting the lower magnitude term NS equation will be

$$\rho_0(\frac{\partial \rho}{\partial t} + u.\nabla u) =- \nabla p + \mu(\nabla^2 u) + \rho g$$

And it also can rewrite as

$$\rho_0(\frac{\partial \rho}{\partial t} + u.\nabla u) =- \nabla p + \mu(\nabla^2 u) + (\rho_0 + \Delta\rho)g$$

Further by changing the density in terms of temperature

$$\rho_0(\frac{\partial \rho}{\partial t} + u.\nabla u) =- \nabla p + \mu(\nabla^2 u) + \rho_0 g - \frac{\rho_0(T-T_0)}{T_0}g$$

And $p = P - \rho_0 gh$

x-Momentum:

$$\rho_0\nabla.(uu) =- \nabla P + \nabla.(\mu\nabla u) + [\rho_0 - \rho_0\beta(T - T_{ref})]$$

$$\nabla.(uu) = - \nabla\overline{p} + \vartheta_{eff}\nabla^2 u + \rho_k g$$

$$\rho_k = 1 - \beta(T - T_{ref})$$

Where,
ρ_k effective kinematic density.

y-Momentum:

$$\rho_0 \nabla.(vv) = -\nabla P + \nabla.(\mu \nabla v) + [\rho_0 - \rho_0 \beta (T - T_{ref})]$$

$$\nabla.(vv) = -\nabla \overline{p} + \vartheta_{eff} \nabla^2 v + \rho_k g$$

z-Momentum:

$$\rho_0 \nabla.(ww) = -\nabla P + \nabla.(\mu \nabla w) + [\rho_0 - \rho_0 \beta (T - T_{ref})]$$

$$\nabla.(ww) = -\nabla \overline{p} + \vartheta_{eff} \nabla^2 w + \rho_k g$$

3.4 Energy equation:

$$\nabla(Tu) = \alpha_{eff} \nabla^2 T$$

$$\alpha_{eff} = \frac{\nu}{Pr} + \frac{\nu_t}{Pr_t}$$

Where
ν is kinematic viscosity
Pr_t turbulent Prandtl number
α_{eff} effective thermal diffusivity

TURBULENT PROPERTY

Turbulence is a highly transient phenomenon, characterized by a wide range of eddy sizes.so here it is not possible to solve all those eddies numerically and obtain a full profile of the turbulent flow field. So a proper turbulence model is required to solve this issue. Here k-ε turbulence model is used. An important feature in turbulence modeling is averaging (like RAS, which is used in this case), which simplifies the solution of the governing equations of turbulence.

Model equations for k-ε

The turbulent ***kinetic energy*** *equation, k*

$$\frac{D}{Dt}(\rho k) = \nabla.(\rho D_k \nabla k) + P - \rho \varepsilon$$

Where ,
k = Turbulent kinetic energy,
D_k = Effective diffusivity for *k,*
P = Turbulent kinetic energy production rate ,
ε =Turbulent kinetic energy dissipation rate,

The ***turbulent kinetic energy dissipation rate*** *equation, ε*

$$\frac{D}{Dt}(\rho\varepsilon) = \nabla.(\rho D_\varepsilon \nabla\varepsilon) + \frac{C_1\varepsilon}{k}(P + C_3\frac{2}{3}k\nabla.u) - C_2\rho\frac{\varepsilon^2}{k}$$

where,
D_ε = Effective diffusivity for *ε*
C_1, C_2 = Model coefficient

The ***turbulent viscosity*** *equation,* ν_t

$$\nu_t = C_\mu \frac{k^2}{\varepsilon}$$

Where,
C_μ = Model coefficient for the turbulent viscosity,
ν_t = Turbulent viscosity,

4. Simulation procedure:

The case deals with 3D turbulent airflow simulation within a negative pressure ventilation room. First, a tutorial case of buoyantPimpleFoam is copied from the tutorial folder and pasted in the desired folder. Then all required input parameters are set before starting the simulation. Also, the blockMeshdict and snappyHexMeshdict files are to be modified as per requirement. A **buoyantPimpleFoam** solver is to be used to run the simulation.

4.1 Creating geometry & Mesh :

- **Geometry**

Room geometry with inlet and outlet is done by using blockMesh. The dimension of the room is 4*4*2.6 m^3 . And for creating the patient and bed model Solidworks software is used. The dimension & geometry of the isolation room, patient bed are given below:

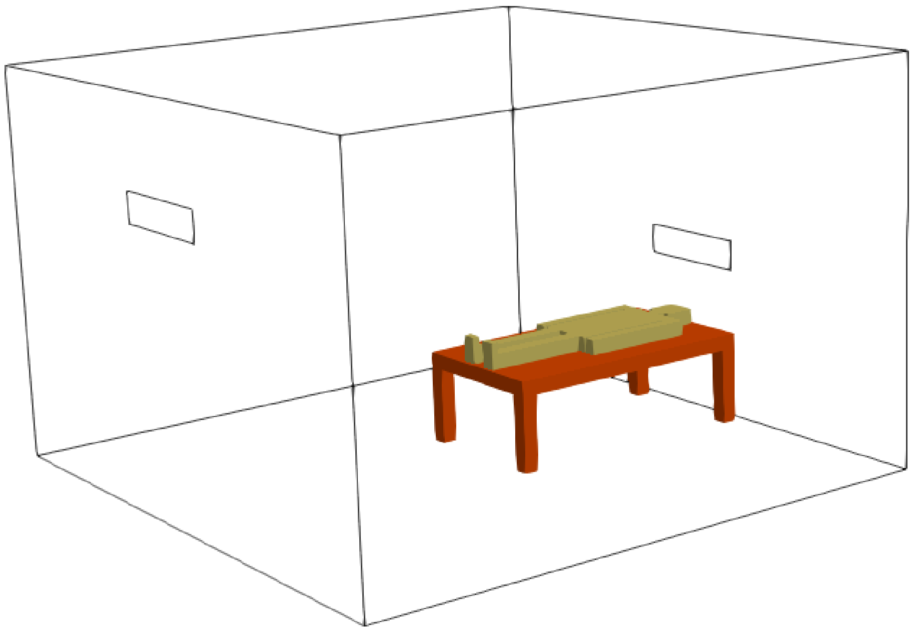


Fig. 3: Geometry of isolation room with bed and patient

Name	X-direction length (m)	Y-direction width (m)	Z-direction Height (m)
Room	4	4	2.6
Bed	1	1.75	0.6
Bed-leg1	0.1	0.1	0.5
Bed-leg2	0.1	0.1	0.5
Bed-leg3	0.1	0.1	0.5
Bed-leg4	0.1	0.1	0.5
Inlet	0.8	0	0.2
Outlet	0.8	0	0.2

Table 2: Dimensions of an isolation room & bed

Name	X-direction length (m)	Y-direction width (m)	Z-direction Height (m)
Body	0.6	0.75	0.1
Head	0.2	0.2	0.1
Hand	0.1	0.75	0.1
Leg	0.1	0.7	0.1

Table 3: Dimensions of patient

➢ Mesh:

The case directories 0, constant, system are created with appropriate sub-directories. The .stl file in the triSurface directory which itself is in the constant directory. The system directory consists of dictionaries like blockMeshDict, snappyHexMeshDict, surfaceFeatureExtractDict.

blockMesh:

The room size 4m*4m*2.6m is designed by the blockMeshDict utility which consists of 9 blocks and simpleGrading (1 1 1) is used to keep aspect ratio near about 1.

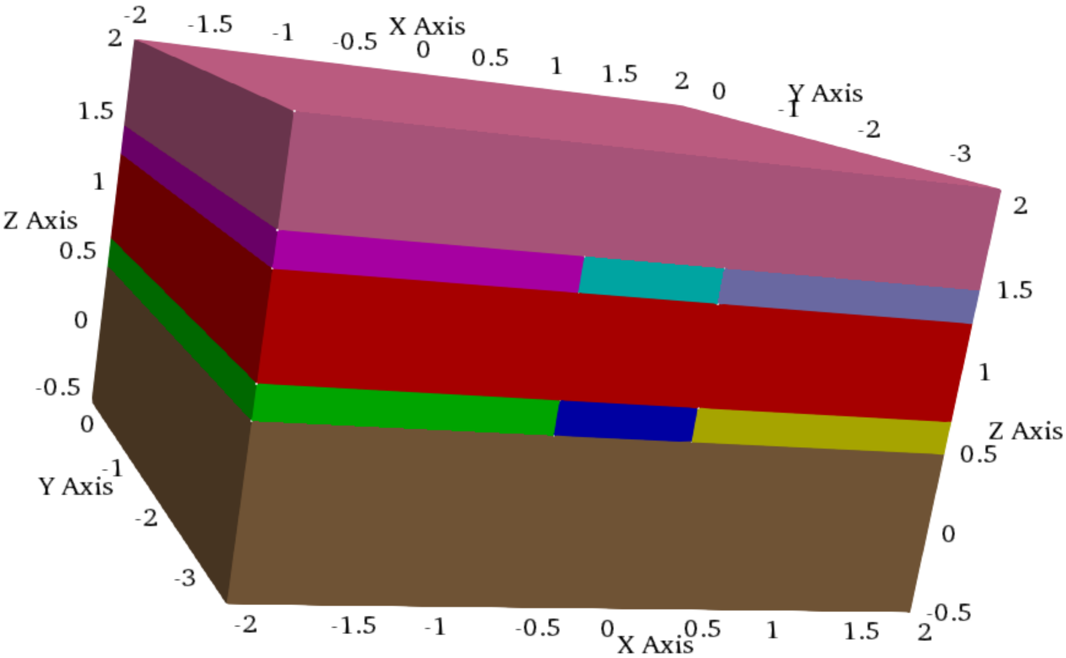


Fig 4 : blocks

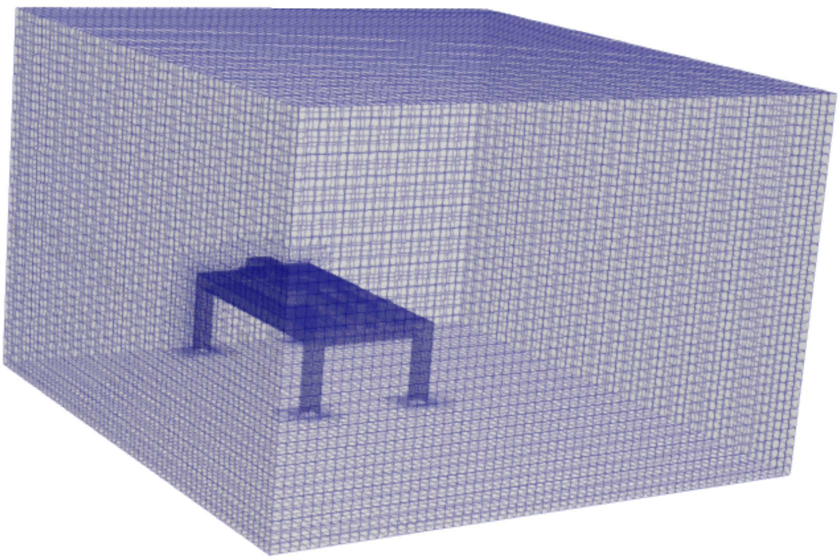


Figure 5: Domain and blockMesh

The snappyHexMeshDict:

The snappyHexMesh is used with the surfaceFeatureExtract utility to mesh the underlying model by following operations:

1) The edges are extracted from the .stl files(bed.stl, patient.stl) through the surfaceExtractFeature and are stored as bed.eMesh and patient.eMesh in the triSurface directory. To run the feature, the surfaceFeatureExtractDict is specified with the .stl files to operate and the settings are tuned to extract all the edges in the .stl files.
2) The domain detects the model and uses castellated Mesh to coarsely remove the blocks that are contained within the model's outline. This is further refined upto the level specified. The refinement surface is set to the eMesh generated, to detect the edges, through which the castellated mesh is generated upon.
3) The snappyHexMesh dictionary is set with refinements of level 2 to capture intricacies near the bed and the patient.

Edge refinement level	2
Surface refinement level	(2 2)

Table 4: SnappyHexMesh refinement

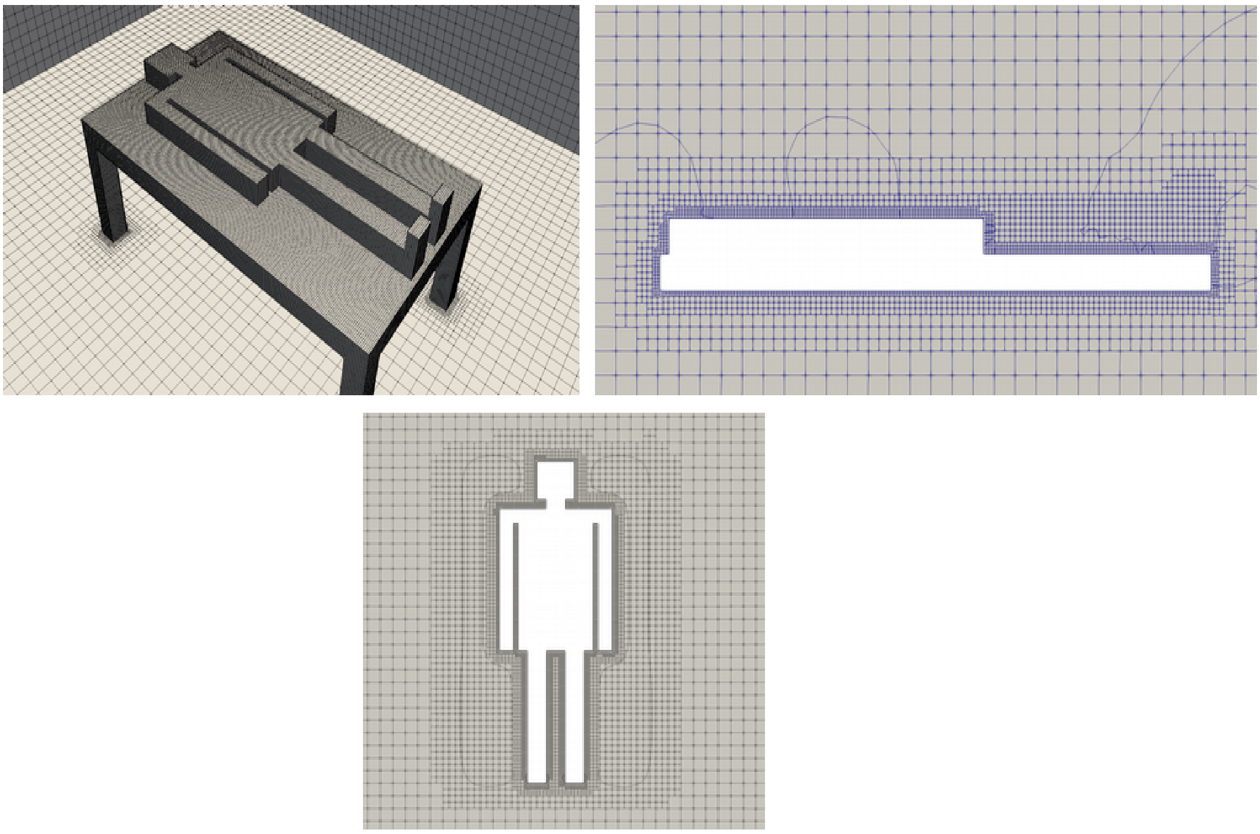


figure 5: Refined Mesh

checkMesh: The mesh is generated and the checkMesh command is run to locate illegal faces of the mesh,if any. The result returned as MESH OK.

Domain	(-2 -3.4 -0.6) (2 0.6 2)
Points	249970
Faces	685635
Internal faces	647373
Cells	217903
Faces per cells	6.11744
Boundary patch	5
Max skewness	1.33146
Max. Aspect ratio	2.72146

Table 5: CheckMesh data

4.2 Initial and Boundary Conditions

There are a total of eight files in the '0' folder. Out of these, four files are for turbulence modelling and the other four files are P_rgh(static pressure), P, T and U.

Boundary	U	P	P_rgh	T
inlet	zeroGradient	calculated	totalPressure $P_0 = 4.5 * 10^{-5}$	fixedValue(295.15K)
outlet	zeroGradient	calculated	fixedValue(-8/-11Pa)	zeroGradient
bed	noSlip	calculated	fixedFluxPressure	zeroGradient
patient	noSlip	calculated	fixedFluxPressure	externalWallHeatFluxTemperature flux(56.52W/ m^2)
wall	noSlip	calculated	fixedFluxPressure	zeroGradient

Table 6: Boundary Condition

4.3 Solver

The solver used is "buoyantPimpleFoam". This is transient solver for buoyant, turbulent flow of compressible fluids for ventilation and heat-transfer.Here Boussinesq approximation is used by changing the equationOfState to Boussinesq in the "thermoPhysicalProperties" file located in "constant" folder to use this solver for the incompressible fluid. The PIMPLE Algorithm is a combination of PISO (Pressure Implicit with Splitting of Operator) and SIMPLE (Semi- Implicit

Method for Pressure-Linked Equations). All these algorithms are iterative solvers but PISO and PIMPLE are both used for transient cases whereas SIMPLE is used for steady-state cases In OpenFoam v-7 “buoyantBoussinesqPimpleFOAM” solver is combined with “buoyantPimpleFoam” solver but approximation can be implemented. Here thermophysical property(thermo type) are shown below for this case to incorporate Boussinesq approximation in this buoyantPimpleFoam solver (for use this solver for a incompressible fluid)

type	heRhoThermo
mixture	pureMixture
transport	const
thermo	eConst
equationOfState	Boussinesq
specie	specie
energy	sensibleInternalEnergy

Table 7:thermoType

4.4 Post-processing

The paraview software can be used to visualize the simulation results in OpenFOAM. This can be run by typing the *paraFoam* command line in the terminal to open the paraview software and upload the case.

5. Results

Ensuring negative pressure inside the whole domain is our foremost priority for which numerous experiments have been conducted to find the most suitable inlet and outlet position so that the chance of the infection of the other people (like doctor, nurse) will be less. So for this, three case studies were performed with different inlet and outlet positions (outlet static pressure -8pa)

Case-1: Validation case
Case-2: Perpendicular inlet-outlet arrangement
Case-3: Parallel inlet-outlet arrangement
Case-4: Improvised solution

Case-1: Validation case

Initially, a referenced case is converted into an OpenFoam case. For that, the same positions of inlet outlets with exact same dimensions are used. Boundary conditions are kept the same and results are analyzed as shown below. Reference paper analyzed is from *Airflow Simulation of an Isolation room using CFD Technique.* [1]

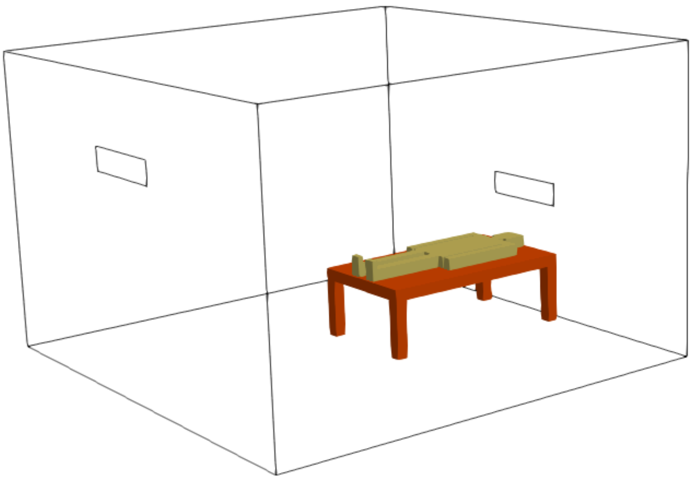


Fig 6: validation case arrangement

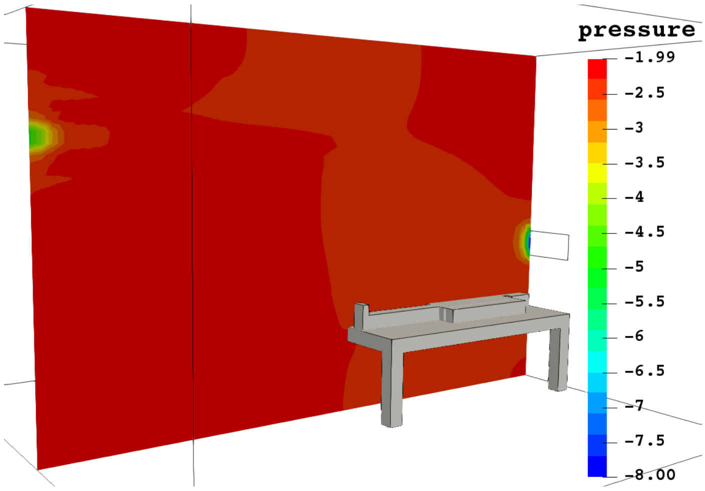


Fig 7: validation case pressure contour

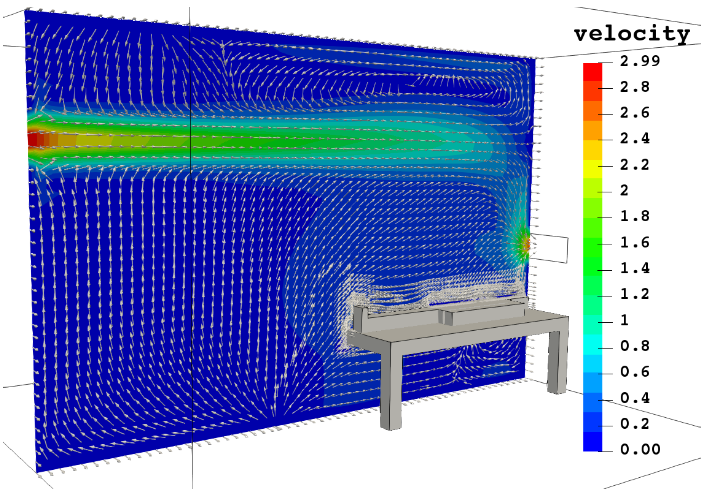


Fig 8: validation case velocity contour & glyph

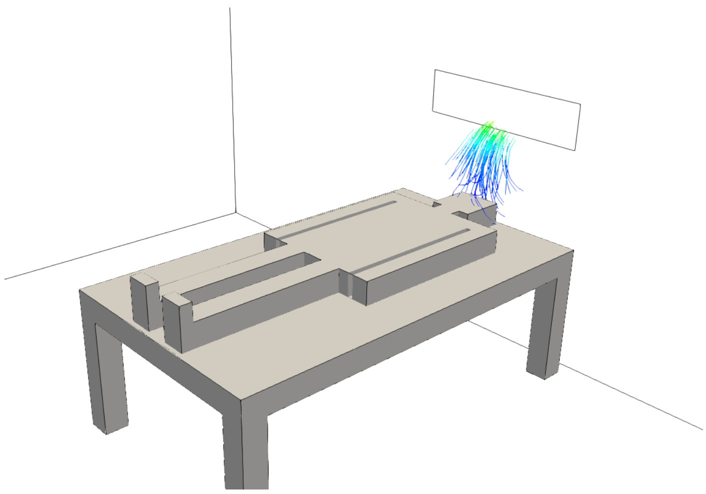


Fig 9: validation case streamline

Case-2: Perpendicular inlet-outlet arrangement

Additionally an perpendicular inlet-outlet arrangement is kept with the same dimensions of the domain. Boundary and Initial conditions kept the same and pressure-velocity contours are analyzed as shown below. Velocity vector glyph and streamlines are shown in the figure.

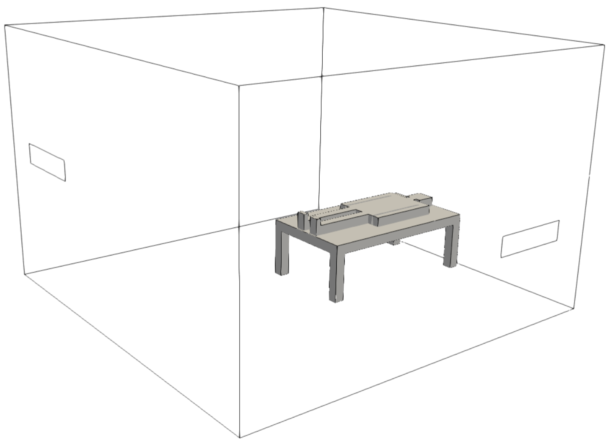


Fig 10: perpendicular inlet-outlet case arrangement

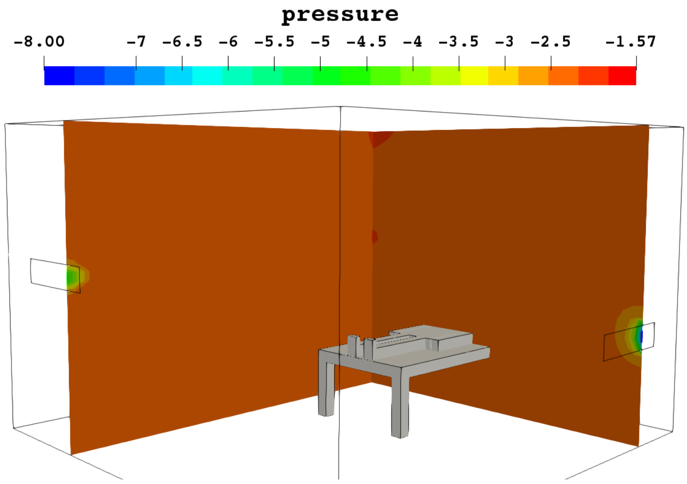


Fig 11: perpendicular inlet-outlet pressure contour

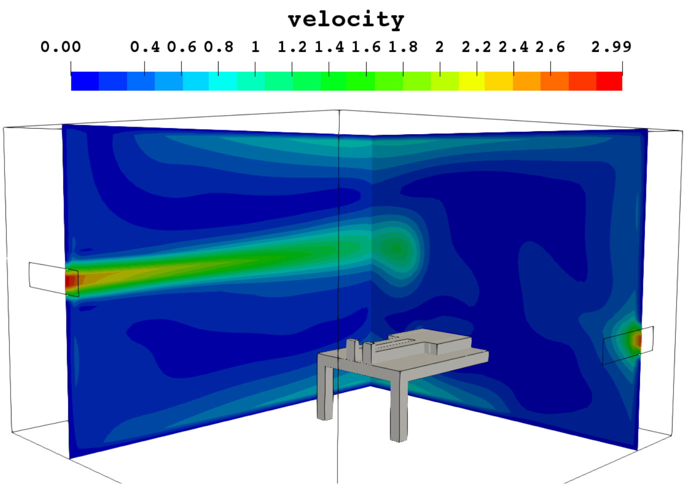


Fig 12: perpendicular inlet-outlet case velocity contour & glyph

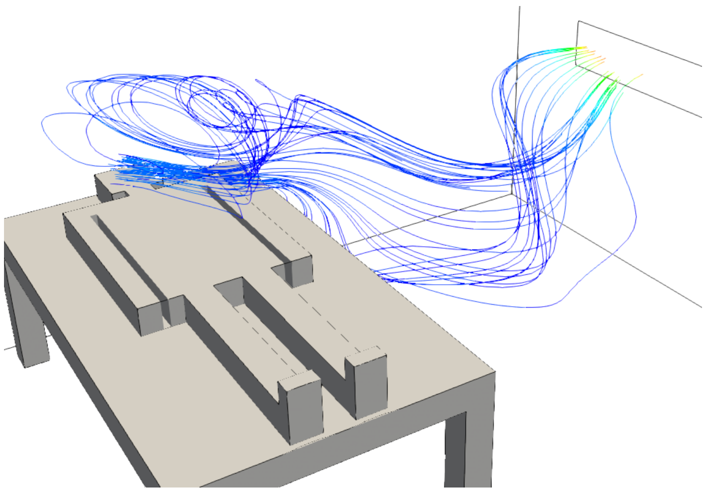
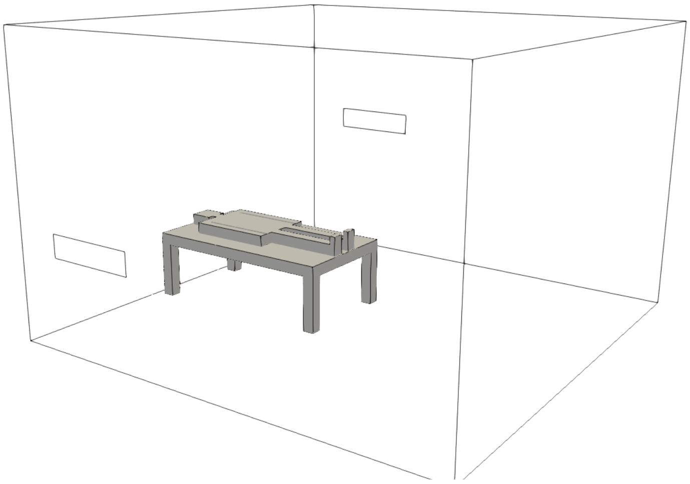


Fig 13: perpendicular inlet-outlet case streamline

Case-3: Parallel inlet-outlet arrangement

Third different inlet outlet position arrangement is shown below in the figure. Results show improvement of air circulation near to the face of the patient. But still there is a scope of improvisation in the case because of the gap in between patient and outlet.



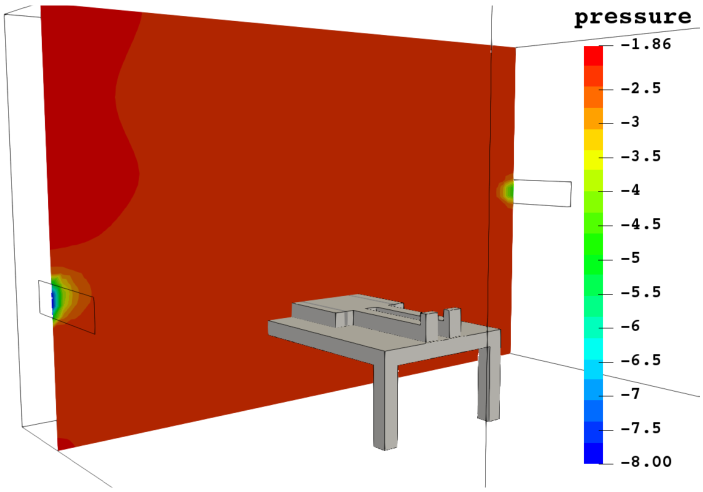
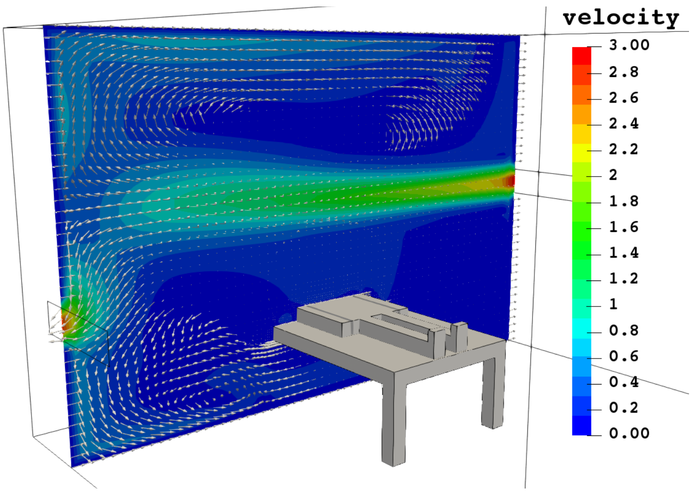


Fig 14: parallel inlet-outlet case arrangement

Fig 15: parallel inlet-outlet case pressure contour



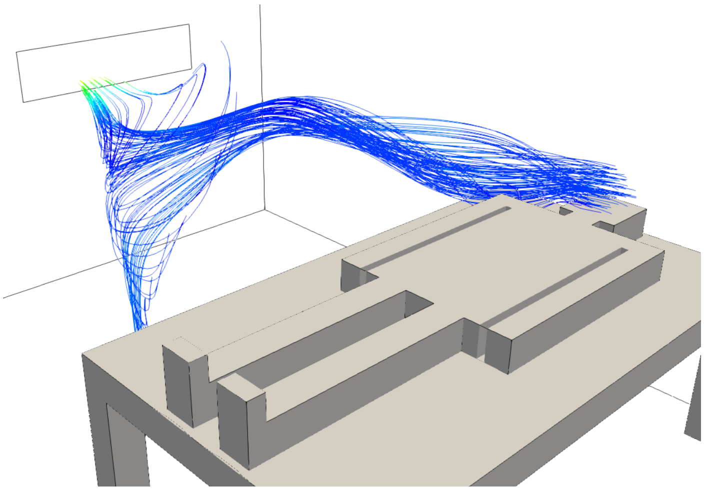
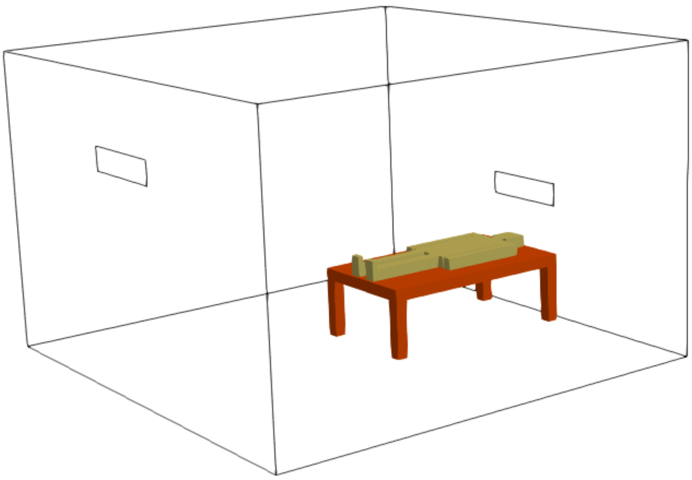


Fig 16: parallel inlet-outlet case velocity contour & glyph

Fig 17: parallel inlet-outlet case streamline

Case-4: Improvised solution

From the above three cases it is observed that the best suitable position of outlet is right above the patient. For better circulation of air near to the patient's head, more negative pressure is applied to the outlet patch. Results show the best outcome of all previous cases.



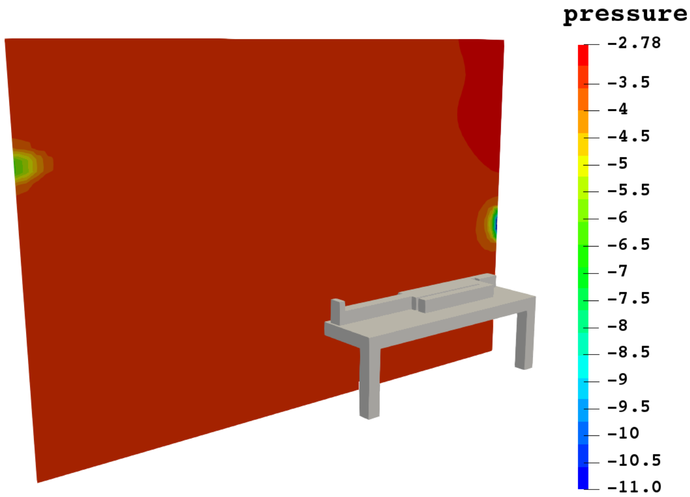
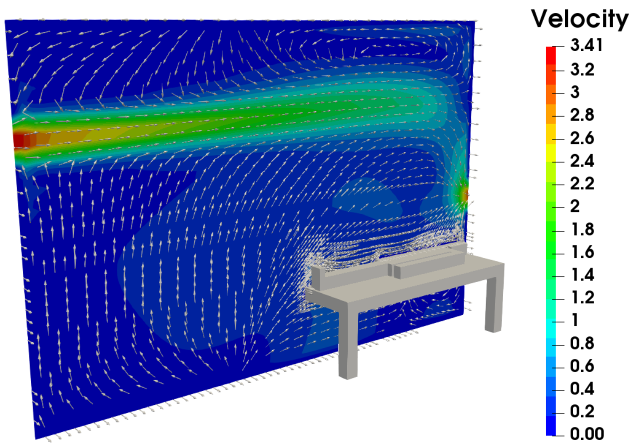


Fig 18: improved case arrangement

Fig 19: improved case pressure contour



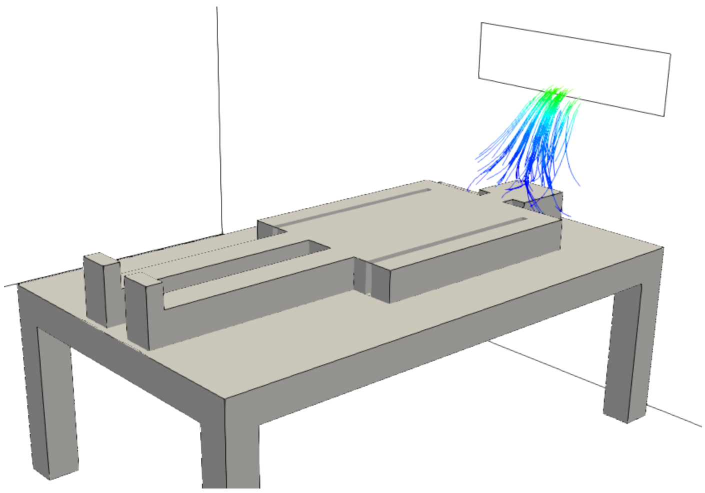


Fig 20: improved case velocity contour & glyph

Fig 21: improved case streamline

Velocity near to the patient's head vs time:

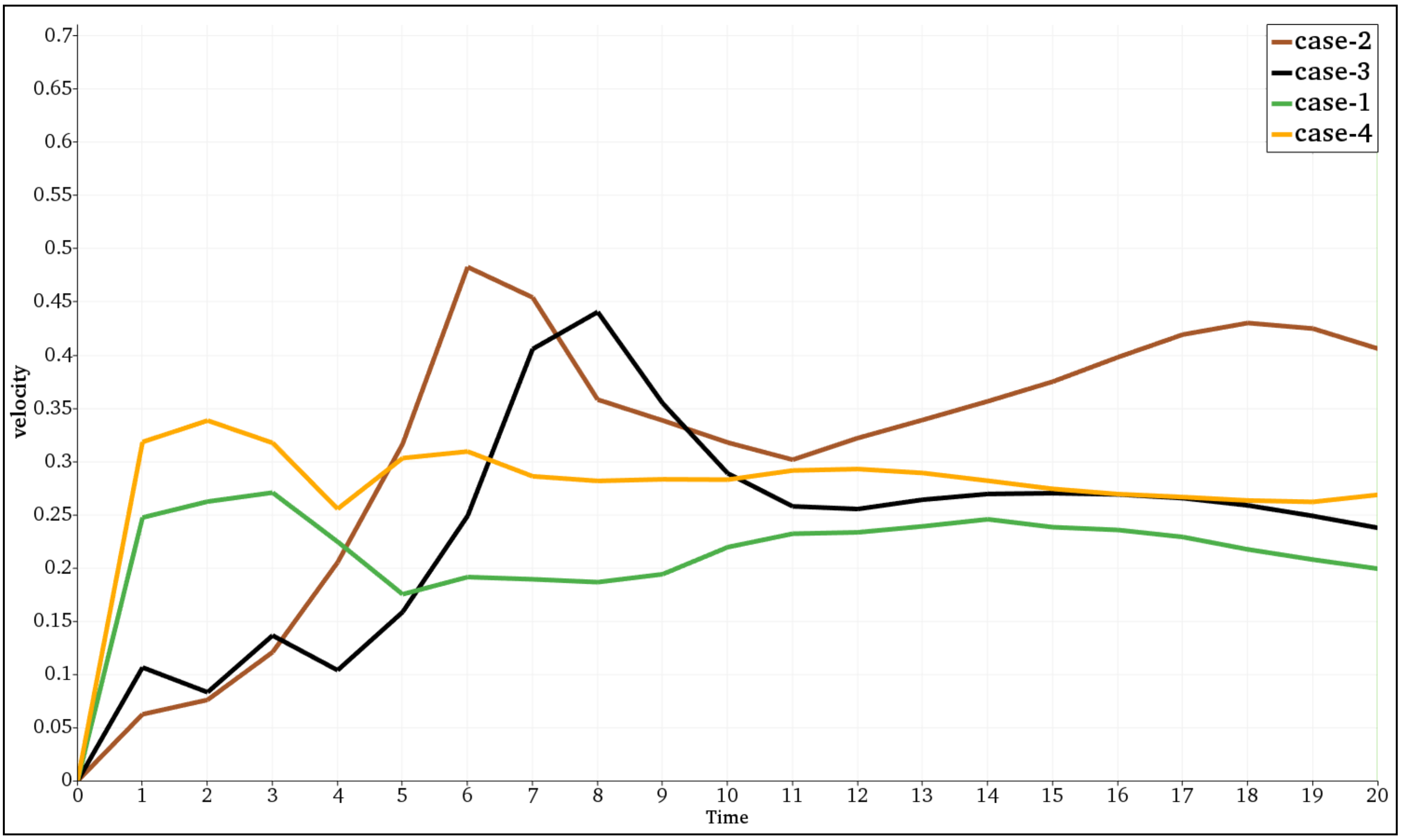


Fig 22: velocity vs time near to the patient's head

6. Conclusion

The focus of this study was to investigate the best inlet-outlet combination for a negative pressure room. First a validation exercise was carried out against reference [7]. Following this different inlet-outlet configurations were tested. From the analysis of the pressure and velocity contours, it is hereby concluded that the best suitable position for inlet and outlet is "Inlet is in the leg side of the patient, outlet is in the head side of the patient" - this position of the room is inherently better than the others due to the fact that the outlet is very near to the bed and hence most of the air which passes over the patient's face goes out of the room through the outlet and there is a very small chance that infected air over the patient's face spreads throughout the entire room. On the other hand for the remaining cases, if any health-care worker (including doctors) stand in between the bed and the outlet then there is a chance of infection of that person due to the contaminated air. That issue can be solved by shifting the bed position near the outlet.

It has also been determined that the most comfortable velocity of < 0.2 m/s ([7]) is satisfied in our simulation case-1, case-3 & case-4.

Reference

[1]. Prasad Mahajan, Arun Saco S, R. Dinesh Kumar, Thundil Karuppa Raj R.Airflow Simulation of an Isolation room using CFD Technique, Volume 118 No. 18 2018, 4261-4269

[2]. Shih, Y.C., Chiu, C.C., and Wang, O., 2007. Dynamic airflow simulation within an isolation room. Building and Environment, 42,3194-3209.

[3]. Chow TT, Ward S, Liu JP, Chan FCK. Airflow in hospital operating theatre—the Hong Kong experience. Proceedings of healthy buildings 2000—design and operation of HVAC systems, vol. 2. 2000, p. 419–24.

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

[5] OpenFOAM User Guide, Version 7

[6] https://www.comsol.co.in/multiphysics/boussinesq-approximation

[7] https://www.simscale.com/blog/2018/12/cfd-airflow-operating-room/

[8] https://en.wikipedia.org/wiki/Thermal_comfort#Elevated_air_speed_method