

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

---

**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  $4*4*2.5\text{ m}^3$ . 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.

## 📖 Technical Data 📖

1. Initial condition of the room will be NTP with zero velocity inside.
2. Air is entering with 22 degree celsius temperature and with negligible velocity.
3. Patient is releasing 58 W/m<sup>2</sup> heat flux.
4. Boundary condition set at outlet in a way that inside room gauge pressure maintains negative.

## 📖 OpenFoam Data 📖

Geometry of Patient and Bed is shown below. Usual dimensions of this type of room are  $4*4*2.5$  meter. SnappyHexMesh utility of OpenFoam is used for final meshing. Buoyancy driven and Boussinesq Approximation based solver called BuoyantPimpleFoam solver is used to carry out simulation.

