Applied CFD in an Indoor environment

Sasan Sadrizadeh

PhD, Docent, Professor

KTH Royal Institute of Technology

Human-Centric Indoor Climate for Healthcare Facilities (HumanIC) HORIZON-MSCA-2022-DN-01

Cohort-Based Training 3 (Jan 2026 19-23)

Learning objectives:

By the end of the lecture, you should be able to:

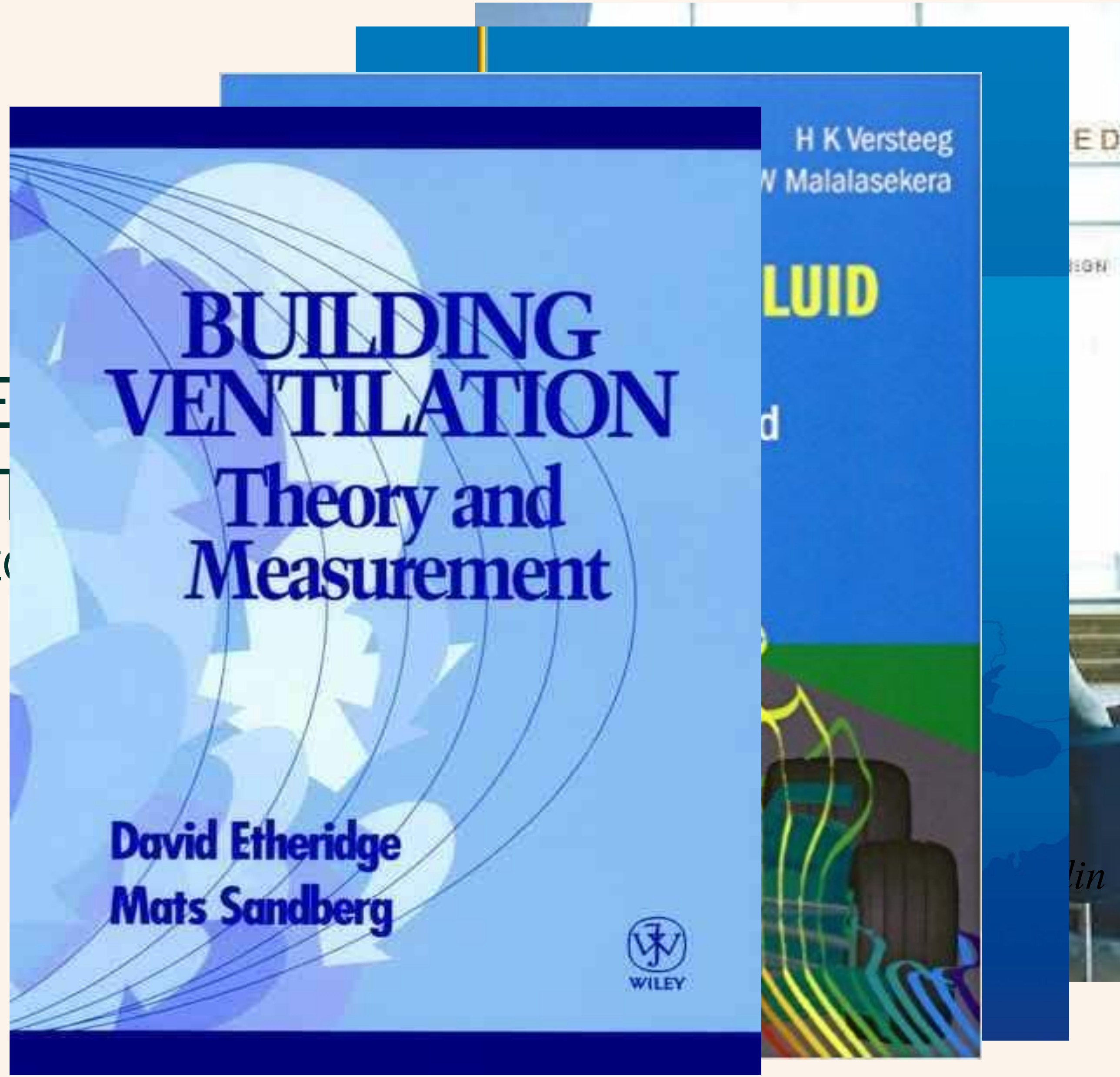
- Explain what CFD is, what equations it solves, and why modeling choices matter.
- Describe the full CFD workflow and identify where errors commonly arise.
- Distinguish CFD from experiments in terms of epistemology, uncertainty, and use.
- Critically assess CFD studies in indoor airflows, ventilation, and contaminant transport.
- Judge when CFD is appropriate and when it is not.

ACHIEVING THE

INDOOR CLIMAT

ENERGY EFFICIENCY ASPEC

by Per Erik Nilsson



What CFD actually is?

Introduction to Computational Fluid Dynamics (CFD):

- CFD is a branch of fluid mechanics that uses computers to solve problems involving **fluid flow**.
- It acts as a middle ground between **analytical** methods (manual equations) and **experimental** methods (physical testing in tanks).
- CFD allows for a **detailed analysis of airflow**, temperature, and pollutant distribution within indoor spaces.
- CFD is a powerful tool for predicting:
  - ✓ Airflow behavior
  - ✓ Heat conduction and Thermal comfort
  - ✓ Pollutant level and distribution
- ANSYS Fluent, StarCCM+, COMSOL Multiphysics, EnergyPlus, etc

Indoor Climate Modelling

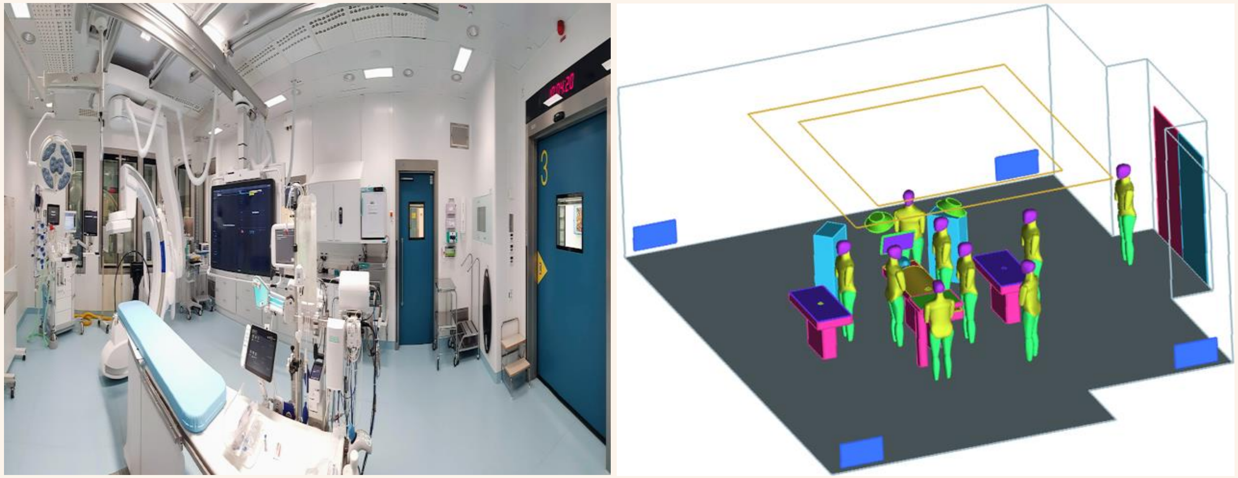
- Why to use CFD:
  - The **complexity of indoor airflow**, particularly in sensitive areas make experimental investigation challenging and costly.
  - **Advances in computer technology** have elevated CFD as a potent alternative for analyzing airflow in enclosed spaces.
  - CFD, a subset of fluid mechanics, is utilized extensively to assess and **visualize air movement**, temperature distribution, and particle concentrations in indoor environments.
  - CFD facilitates **valuable predictions concerning** ventilation system parameters such as **air velocity**, **temperature**, and **humidity**.
  - Acquiring precise data through CFD provides a **solid foundation** for making informed **design decisions**, proving instrumental in optimizing indoor climate and ventilation systems.

The Mechanics of CFD

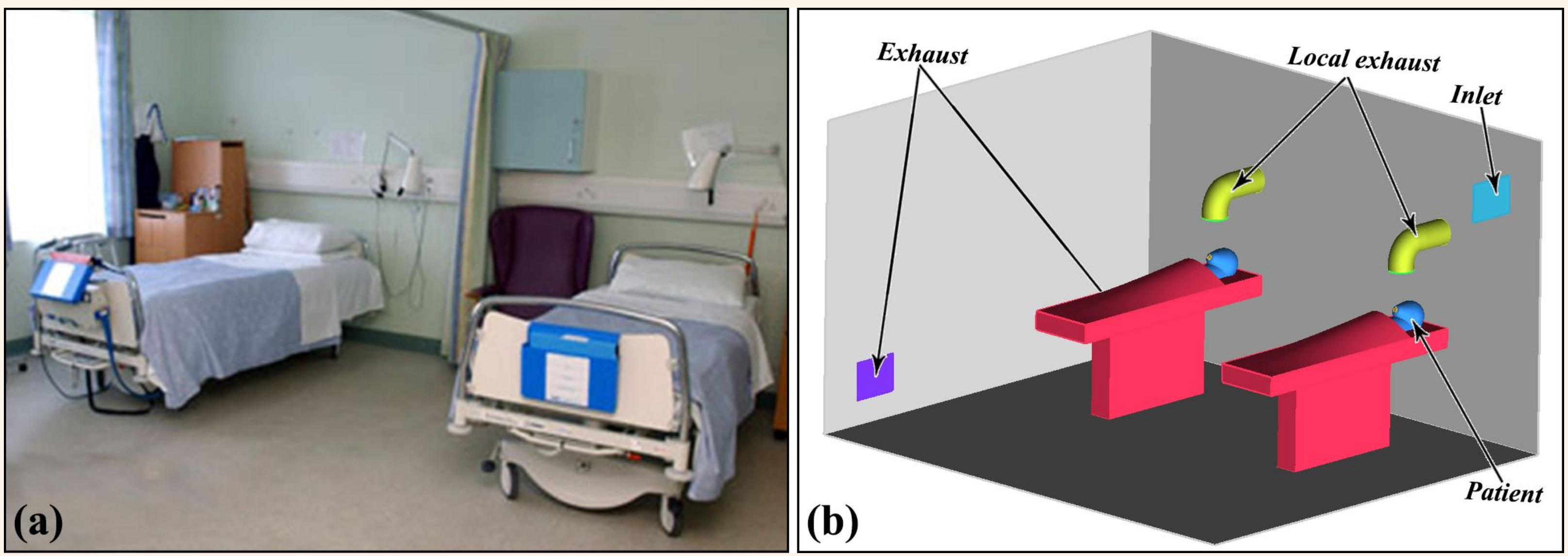
- The steps in CFD simulations are typically as follows:
  - **Pre-processing:** This step involves defining the problem geometry, meshing the geometry, and setting up the boundary conditions.
  - **Solver:** This step involves selecting a solver and setting up the simulation parameters.
  - **Simulation:** This step involves running the simulation and monitoring the results.
  - **Post-processing:** This step involves analyzing the simulation results and generating visualizations.

Steps of a CFD simulation

- Pre-processing:
  - **Geometry:**
    - ✓ Defining the geometrical configuration (computational domain) of the problem.
    - ✓ Employ simplification in complex geometries to concentrate on crucial areas.
    - ✓ In scenarios necessitating detailed analysis of flow patterns or particle distribution near objects or individuals, opt for realistic feature models.



Danderyd Hospital PCI Heart Surgery OR and its geometry replica

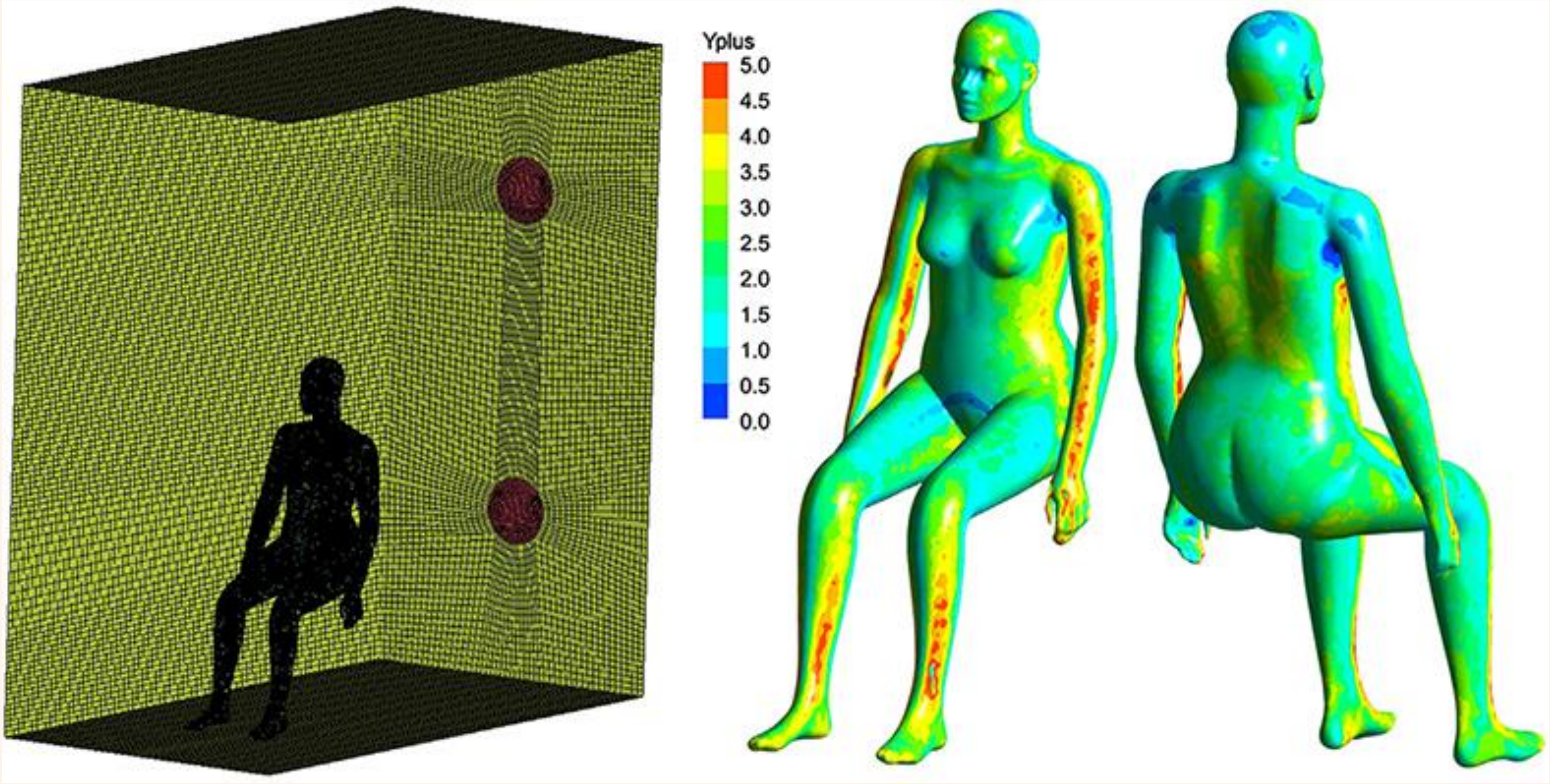


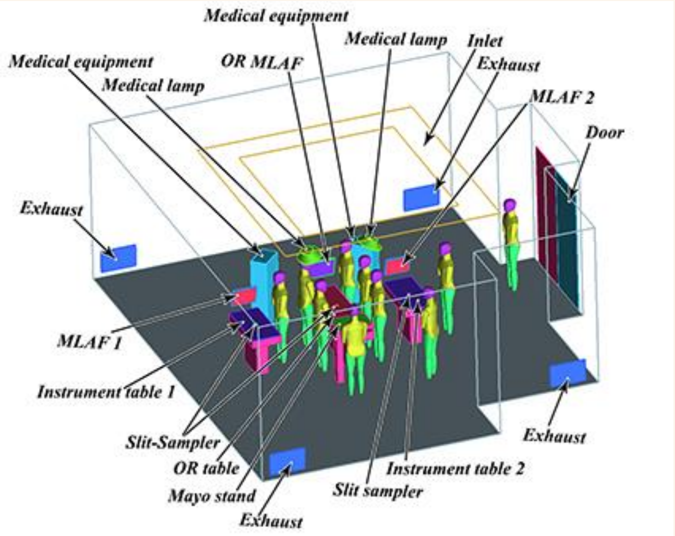
Two-bed hospital ward (a) and its model in CFD simulation (b)

Steps of a CFD simulation

- Pre-processing:
  - **Meshing:**
    - ✓ The fluid-occupied space is subdivided into discrete cells known as a grid or mesh.
    - ✓ Optimal grids are finer in areas of large flow field variation and coarser in stable regions.
    - ✓ While finer grids enhance accuracy, they demand more simulation time and computing power due to increased element numbers.
    - ✓ To mitigate grid resolution effects on results, grid-independent test is crucial for high-quality CFD studies.

2. Mesh: Near-wall resolution, y^+ concepts:





Steps of a CFD simulation

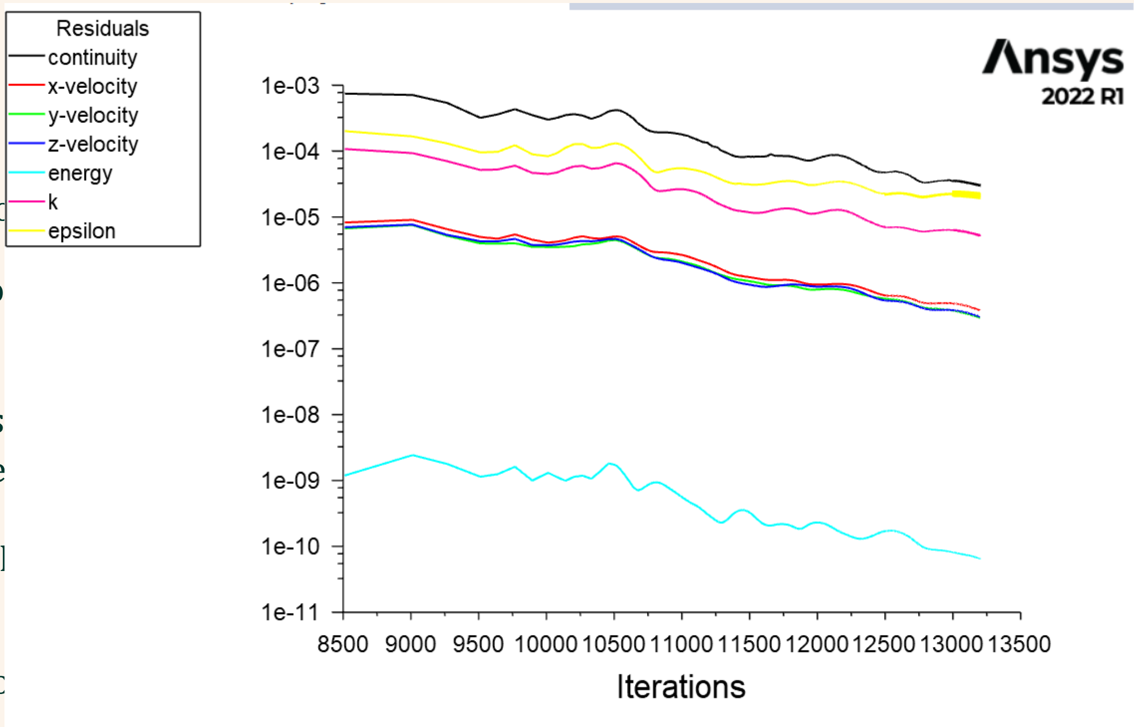
- Pre-processing:
  - **Boundary Conditions:**
    - ✓ Establish the physics that needs to be included to calculate the airflow and properties.
    - ✓ Common types include *inlet*, *outlet*, *wall*, and *symmetry* conditions. Each type dictates how fluid behaves at the boundary, e.g., specifying velocity at the inlet or pressure at the outlet.
    - ✓ Properly set boundary conditions significantly impact the accuracy and reliability of the simulation results, affecting the overall quality of the analysis and the insights derived from it.

Steps of a C

Navier-Stokes equation

$$\rho\left(\frac{\partial u}{\partial t} + (u \cdot \nabla)u\right) = \rho g - \nabla p + \mu \Delta u$$

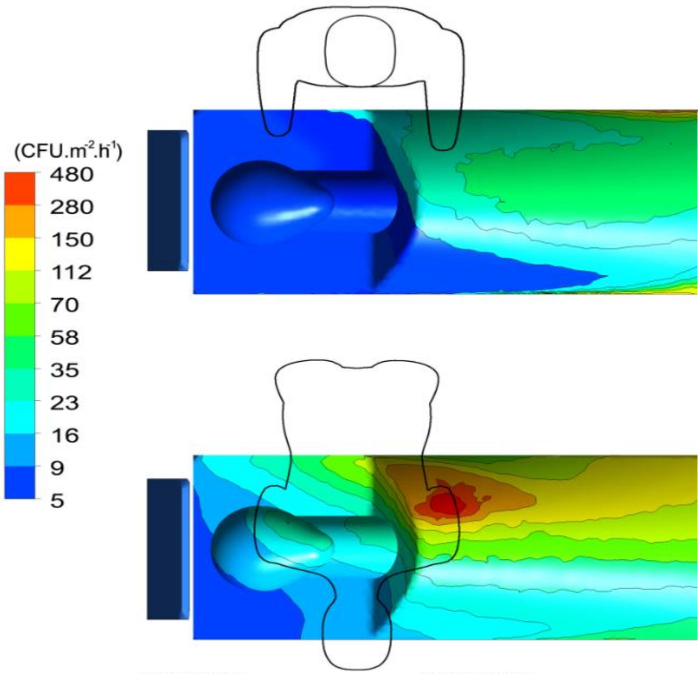
- Processing:
  - **Solve:**
    - ✓ The next step in a CFD simulatic
    - ✓ The solver is the numerical algo flow.
    - ✓ The fundamental basis of almos mathematical description of the
    - ✓ Once the solver and simulation **simulation.**
    - ✓ There is no general analytical sc find specific solutions

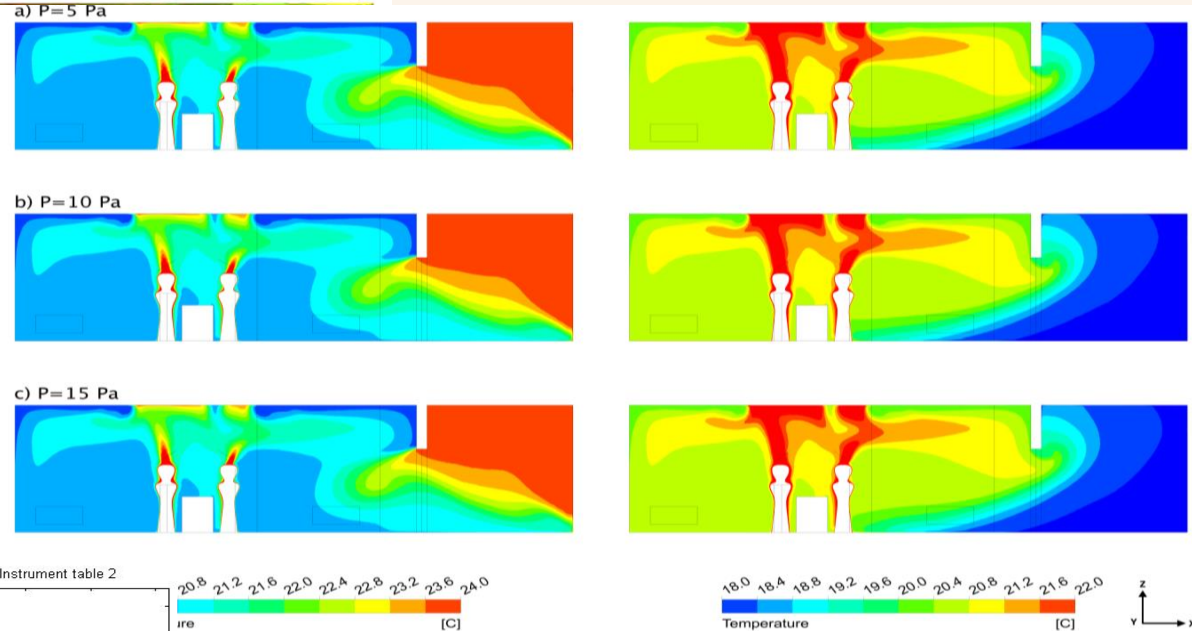


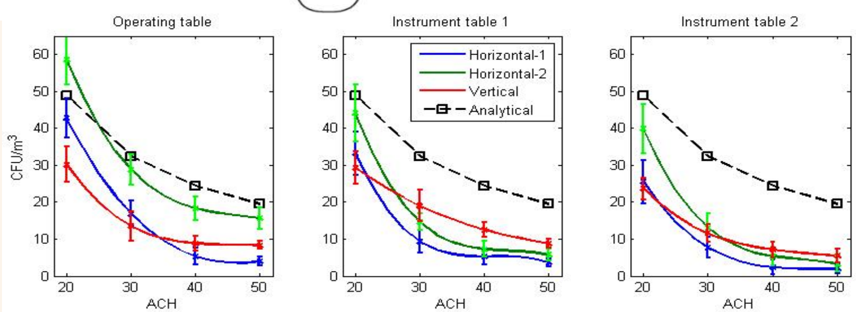
    - ✓ The solver will **iterate until the convergence** criteria are met. This means the solution has reached a state that does not change significantly from iteration to iteration.

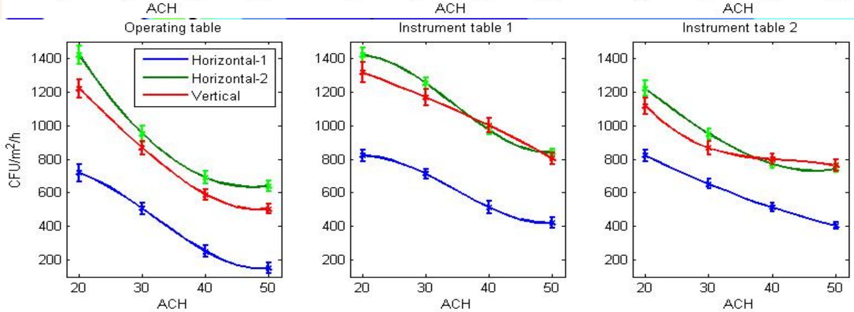
Steps of a CFD simulation

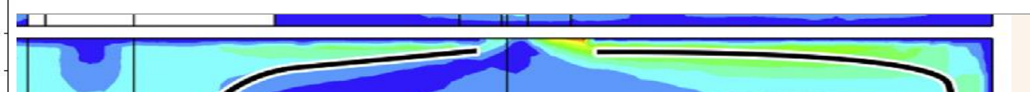
- Post-Processing:
  - **Result visualization and data analysis:**
    - ✓ Post-processing is a crucial step following CFD simulations to derive insights from fluid flow and particle dispersal predictions.
    - ✓ It shows how fluid interacts with every part of a geometry, such as air flowing around a body or walls. How contaminants spread in the environment, …
    - ✓ Various visualization techniques like animation, streamlines, vector plots, line and contour plots, and 2D and 3D surface plots are used for comprehensive analysis.
    - ✓ Dynamic results are often illustrated using particle tracking and animation, enhancing understanding and interpretation of simulation outcomes.

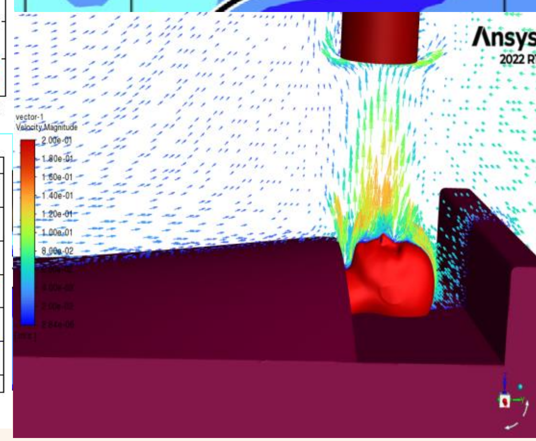

(CFU.m-2.h-1)
480
280
150
112
70
58
35
23
16
9
5

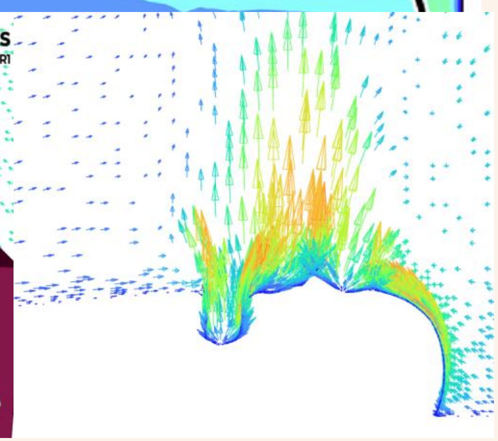

a) P=5 Pa
b) P=10 Pa
c) P=15 Pa
20.8 21.2 21.6 22.0 22.4 22.8 23.2 23.6 24.0
[C]
18.0 18.4 18.8 19.2 19.6 20.0 20.4 20.8 21.2 21.6 22.0
Temperature
[C]


Operating table
Instrument table 1
Instrument table 2
Horizontal-1
Horizontal-2
Vertical
Analytical
CFU/m3
ACH


Operating table
Instrument table 1
Instrument table 2
Horizontal-1
Horizontal-2
Vertical
CFU/m2/h
ACH




Ansys
2022 R1



Model Validation

- ✓ Assessment of model predictive capability by comparison with independent experimental data.
- ✓ Focuses on relevant output quantities, not full-field agreement.
- ✓ Requires faithful reproduction of geometry, boundary conditions, and operating conditions.
- ✓ Agreement is context-specific; a model validated for one flow regime is not universally valid.
- ✓ Validation does not prove correctness; it establishes credibility within defined limits.
- ✓ Typical acceptance criteria (e.g., ±5%) are application- and metric-dependent, not universal.
- ✓ Experimental **uncertainty must be quantified** before judging CFD accuracy.

Verification vs Validation

Validation: assesses whether the chosen physical and mathematical model accurately represents reality for a given application by comparing it with experimental data.

Verification: Are the equations solved correctly? assesses whether the numerical implementation accurately solves the chosen equations.

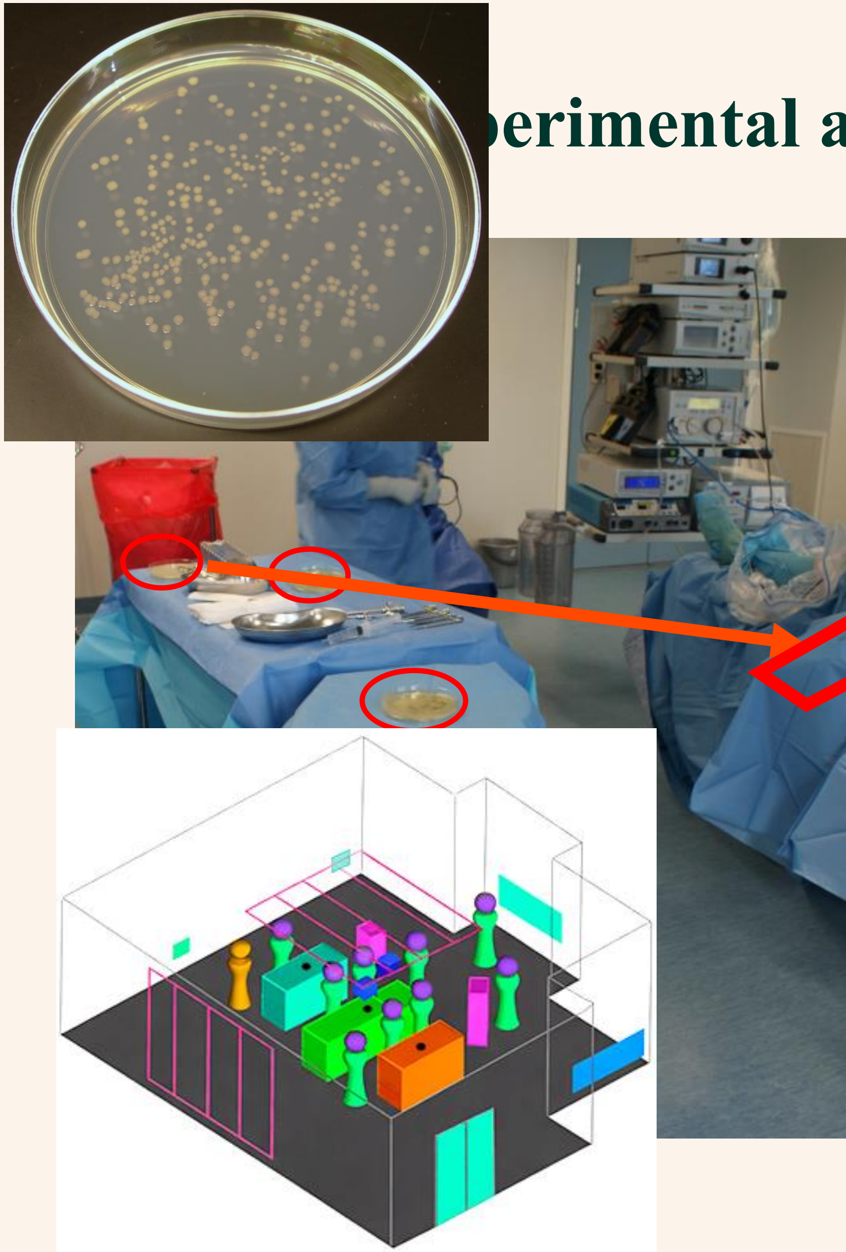
What verification cannot tell you

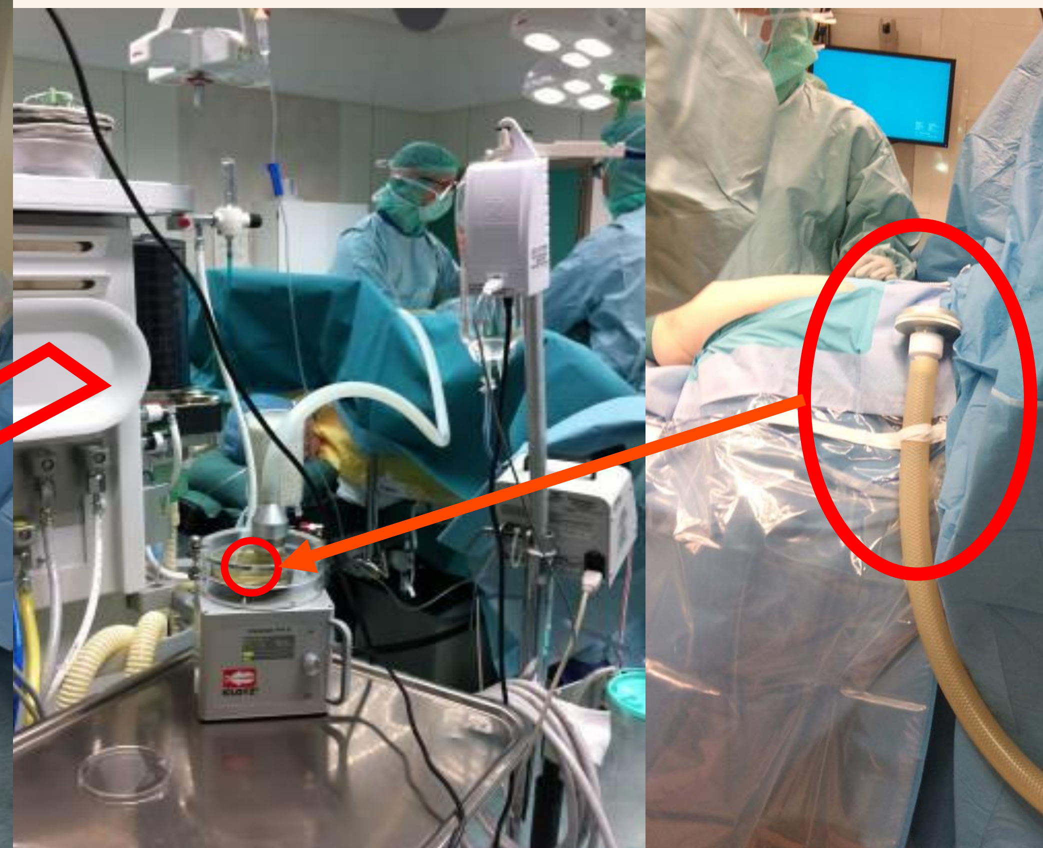
- Whether the turbulence model is appropriate.
- Whether buoyancy is modeled realistically.
- Whether the inlet turbulence level is physically correct.
- Those are validation questions.

In other words:

- ✓ **Validation** asks: Is this the right model?
  - By comparing CFD to reality (experiment).
- ✓ **Verification** asks: Is this model solved right?
  - By comparing CFD to CFD expectations.

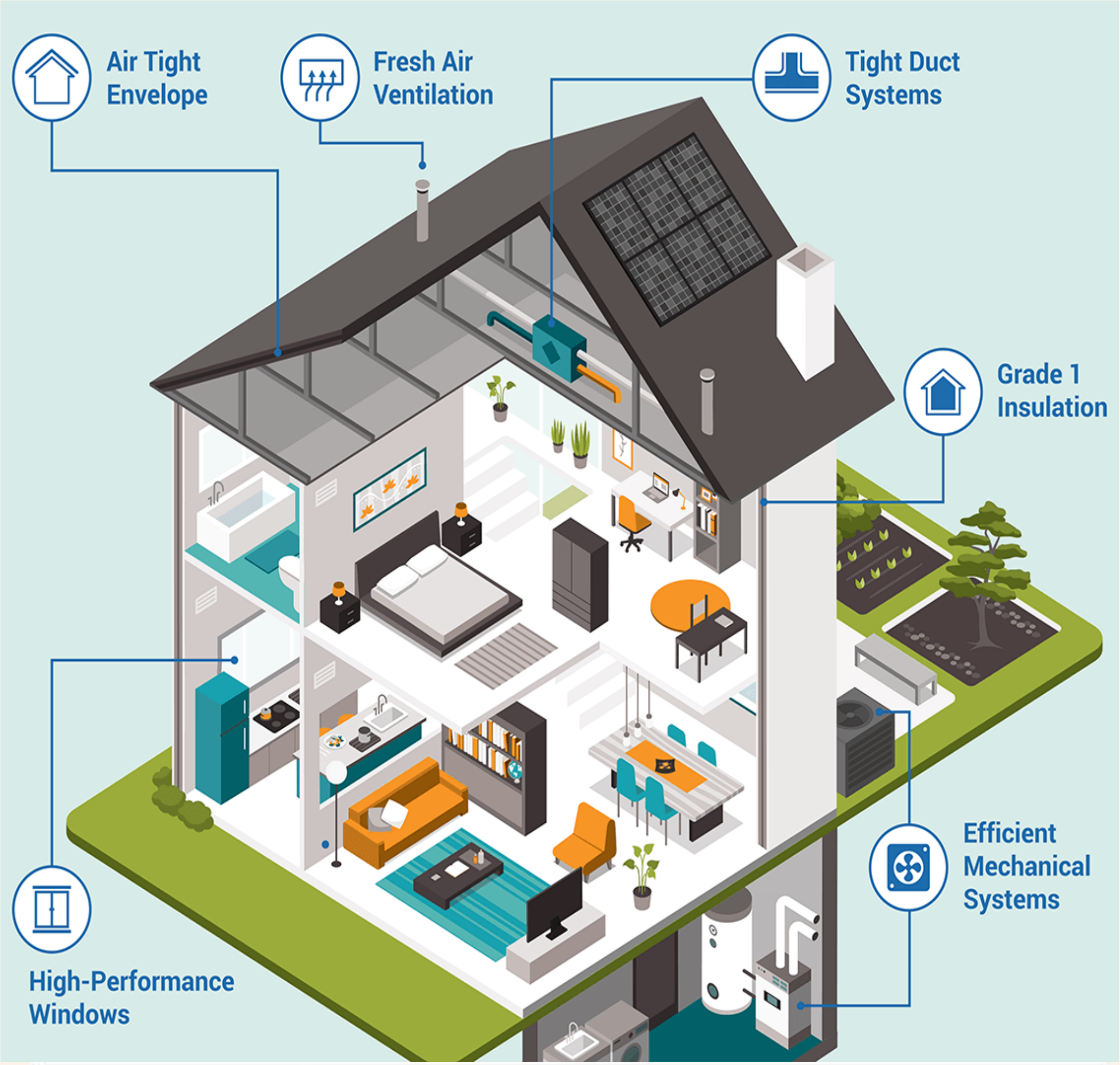
perimental and Numerical analysis





Overview

- Indoor Air Quality (IAQ)
- Air distribution in Rooms
  - ➢ Piston Flow
  - ➢ Displacement Flow
  - ➢ Mixing Flow
- Thermal Comfort
- Indoor Climate Modeling
  - ➢ The numerical modeling
  - ➢ Computational Fluid Dynamics



Air distribution in Rooms (Ventilation)

- Provide good indoor air quality (IAQ)
- Maintain acceptable thermal comfort
- Fresh air is evenly distributed in room
  - Fresh Air = Outdoor Air
  - Fresh Air = Conditioned Air
- Removing contaminants
  - Volatile organic compounds (VOCs)
  - Pollutants
  - CO_2 level

Air distribution in Rooms (Ventilation)

- Airflow Patterns
  - Laminar
  - Turbulent
- Fresh air is evenly distributed throughout the room
  - Placement of supply and return vents
  - Even distribution and prevent stagnant air zones
- Ventilation Rate and Air Exchange
  - Appropriate ventilation rate for the specific room and its occupancy
  - Dilute (or displace) indoor pollutants and maintain acceptable IAQ
  - Air exchanges per hour (ACH) is a key metric used to quantify ventilation effectiveness

$$\text{ACH} = \frac{\text{Volume of Air Flow (in Cubic Meter Per Hour)}}{\text{Volume of the Area}}$$

ACH: Air Changes Per Hour

Air distribution in Rooms (Ventilation)

- Air exchanges per hour (ACH)
  - Appropriate ventilation rate for the specific room and its occupancy

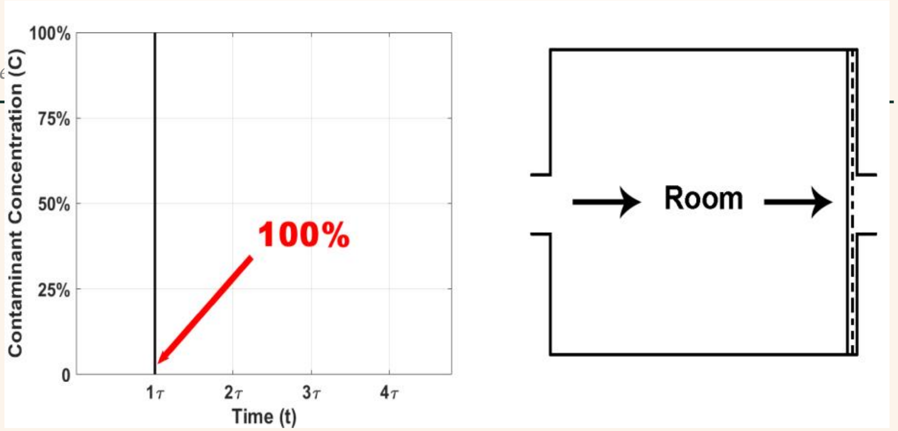
$$ACH_{\text{Air Changes Per Hour}} = \frac{Q_{[m^3/h]}}{V_{[m^3]}} = \frac{1}{h}$$

$$ACH = 2h^{-1}$$

- The nominal time constant (τ)
  - Ratio of the domain volume to the supply air volume to that domain

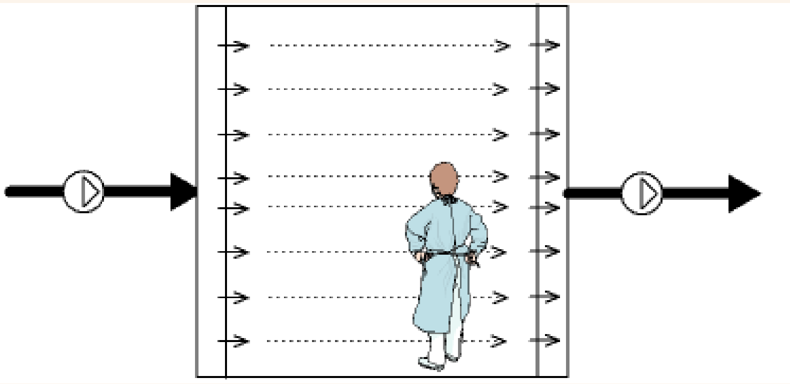
$$\tau_{\text{Nominal time constant}} = \frac{V_{[m^3]}}{Q_{[m^3/h]}} = h$$

Sasan Sadrizadeh (ssad@kth.se

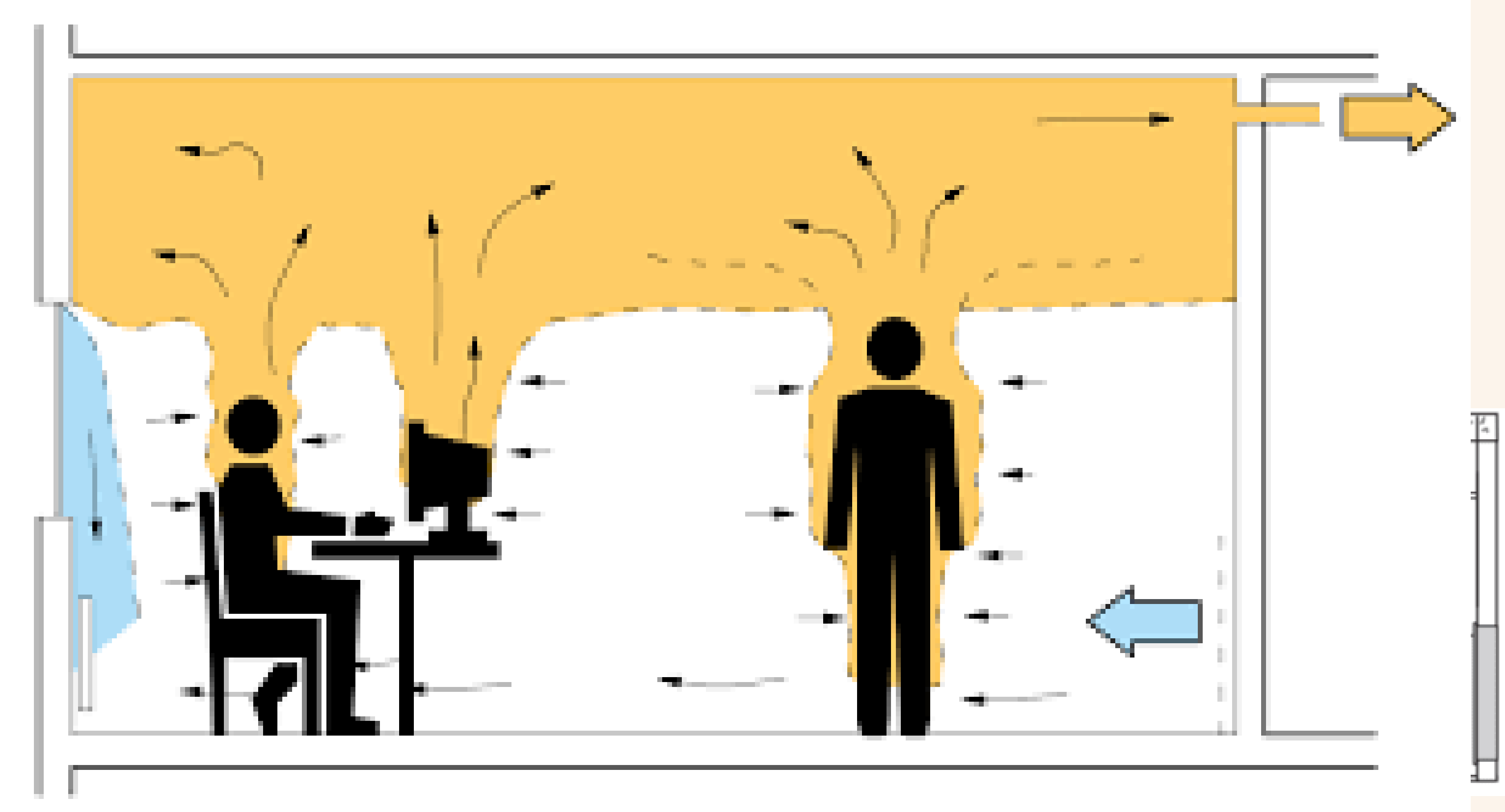


Piston Flow (Plug Flow)

- Air move in **parallel** paths along the length of a room or channel with **no lateral mixing**.
- The air **velocity remains constant** across any cross-section perpendicular to the flow direction.
- The fluid elements at the **inlet reach the outlet** before those entering later, maintaining a first-in, first-out sequence.
- Often used in engineering analyses to simplify complex flow behaviors, particularly useful in **reactor design and clean rooms**.
- Real-world deviations may introduce elements of **dispersion or turbulent flow**.



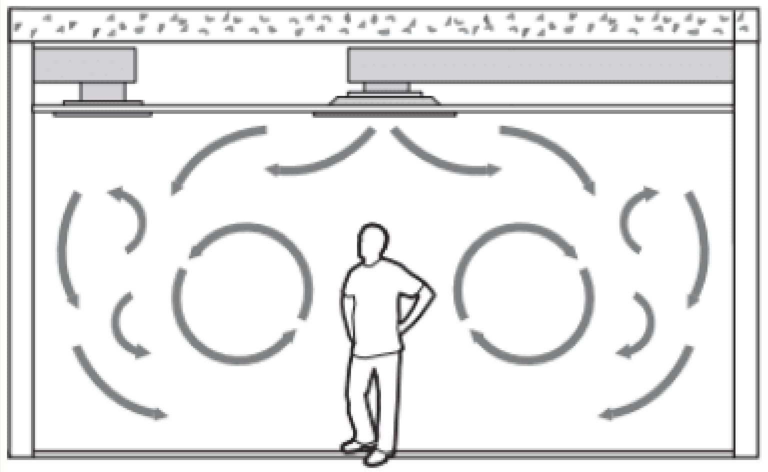
Displacement Flow (Buoyancy-driven flow)



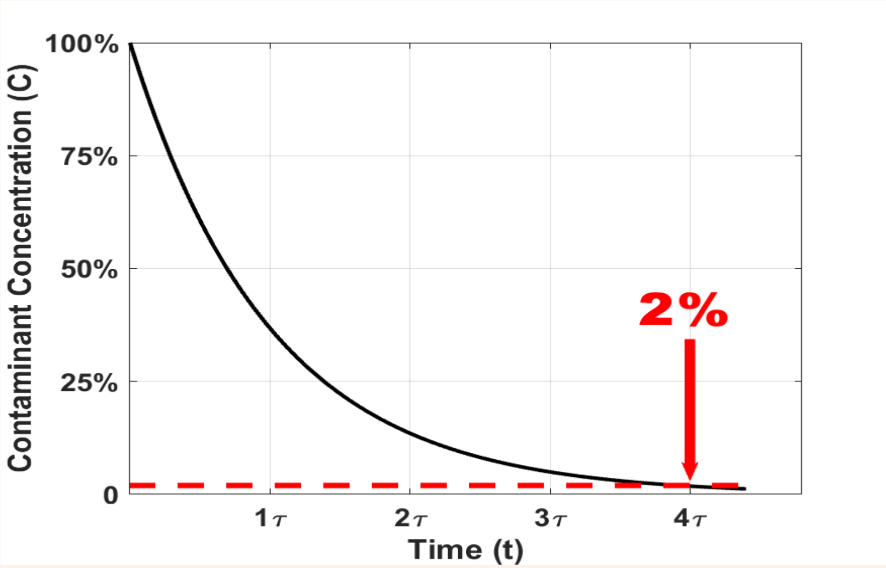
Mixing Flow (fully Mixing Flow)

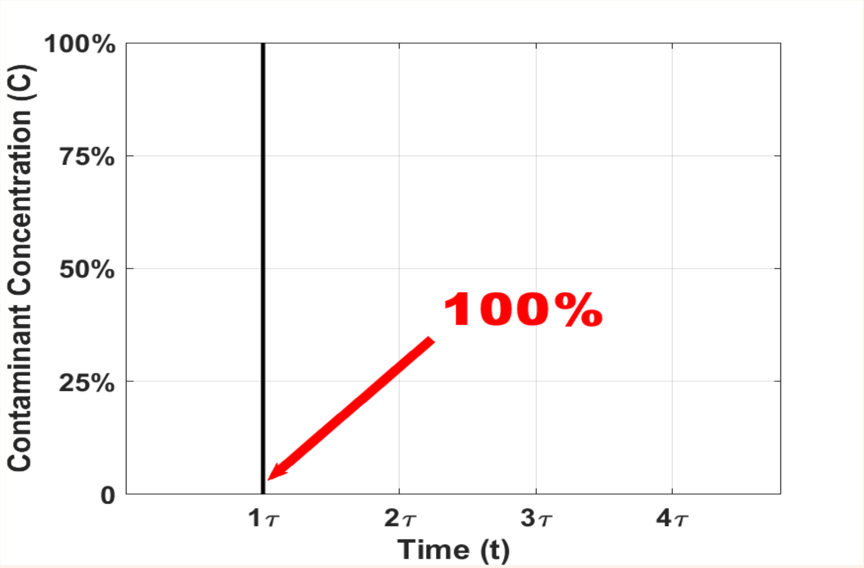
- Rely on the **delusion concept** to reduce contaminant concentration in the room
- Fresh air is supplied into space and **mixed** with the existing indoor air
- Supply **air jets are high enough** to induce movement in the room air, promoting thorough mixing and helping in homogenizing the temperature
- To achieve uniform conditions in the occupied zone, **minimizing temperature** and **contaminant concentration** gradients

- Commonly used in various settings due to its **simplicity and effectiveness** in maintaining comfort and indoor air quality.



Mixing Flow (fully Mixing Flow)





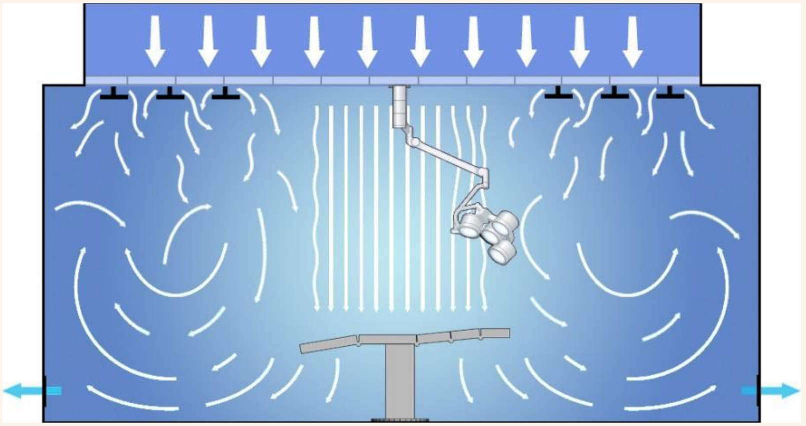
$$C(t) = C_0 . e^{\frac{-t}{\tau_n}}$$

- C_0: initial indoor concentration
- C_t: average indoor concentration after t time
- τ_n: Nominal time constant

Hybrid Ventilation

- Combination of **two or more types** of ventilation systems
- Provides the **best of both worlds** in terms of energy efficiency, comfort, and control
- Can be used to **reduce energy consumption** and **improve indoor air quality**
- Help to create a **more comfortable environment** by reducing drafts and temperature gradients

- Hybrid ventilation systems can be designed to **meet the specific needs of a building** and its occupants. They can be used in various settings, including offices, schools, hospitals, and homes.



Thermal Comfort

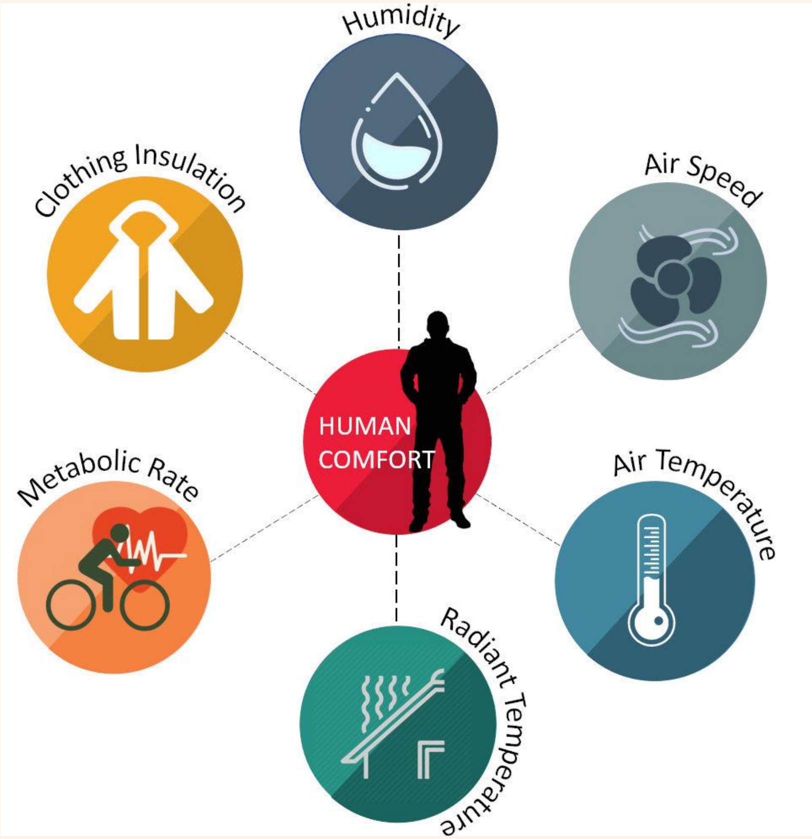
- Thermal comfort is a **state of mind** that expresses satisfaction with the thermal environment.
- It is assessed by **subjective evaluation** (ASHRAE Standard 55)

- The optimal thermal comfort conditions vary depending on the **individual** and the **activity level**.
- There are a number of ways to create a comfortable thermal environment, such as through **building design**, **HVAC systems**, and personal **clothing choices**.
- Achieving thermal comfort is crucial for the **well-being** and **productivity** of occupants in indoor environments.

Thermal Comfort

Factors Influencing Thermal Comfort:

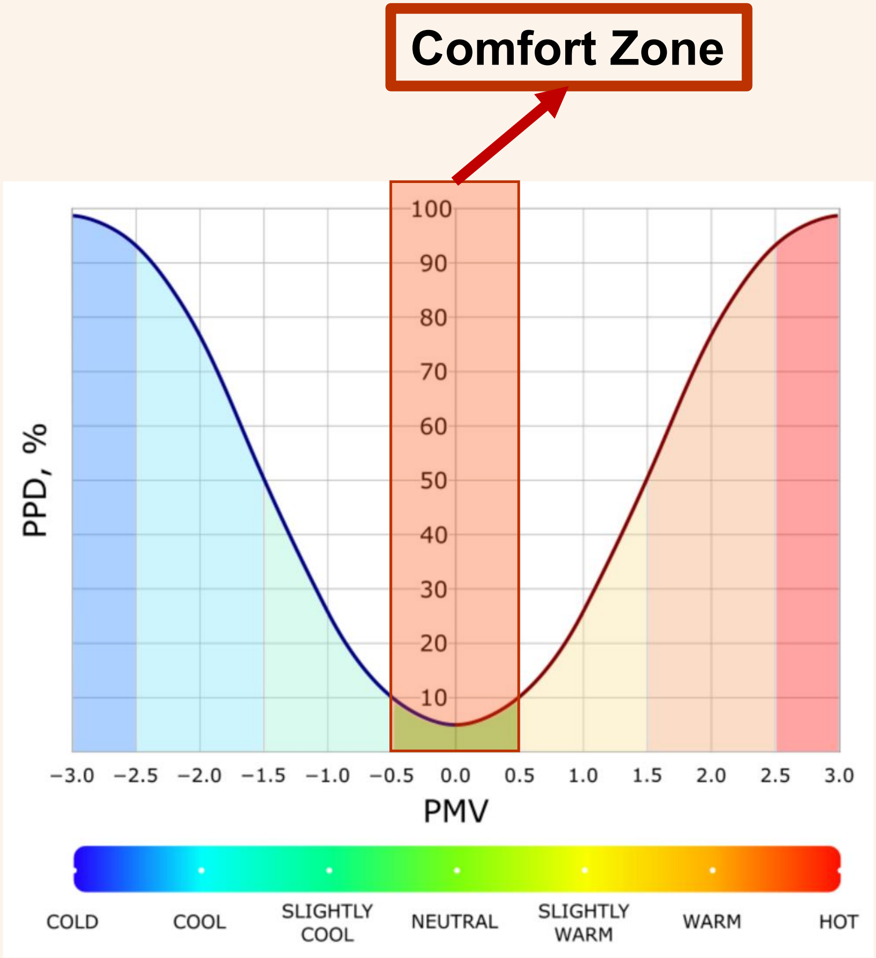
1. **Metabolic Rate:** The energy generated by the human body.
2. **Clothing Insulation:** The level of insulation provided by clothing.
3. **Humidity:** The moisture content in the air.
4. **Air Speed:** The speed of air moving across the body.
5. **Air Temperature:** The temperature of the air surrounding the body.
6. **Radiant Temperature:** The temperature of surrounding surfaces.



Thermal Comfort

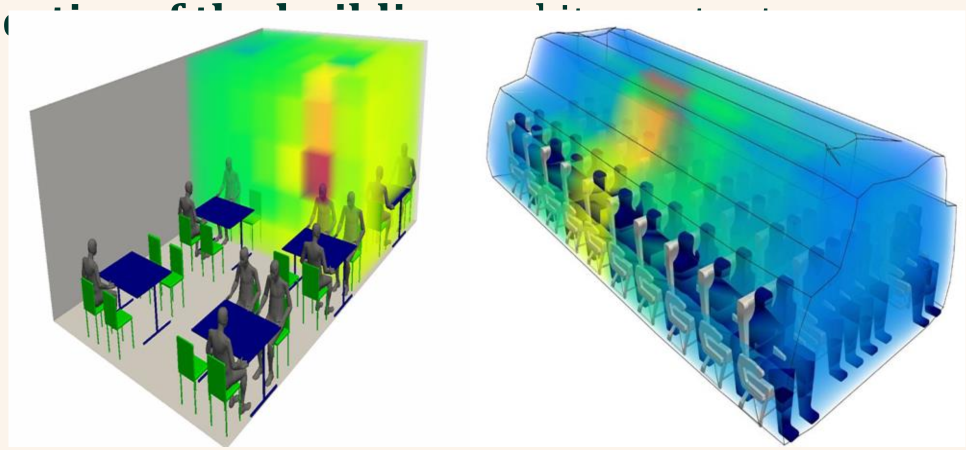
Assessing and Enhancing Thermal Comfort:

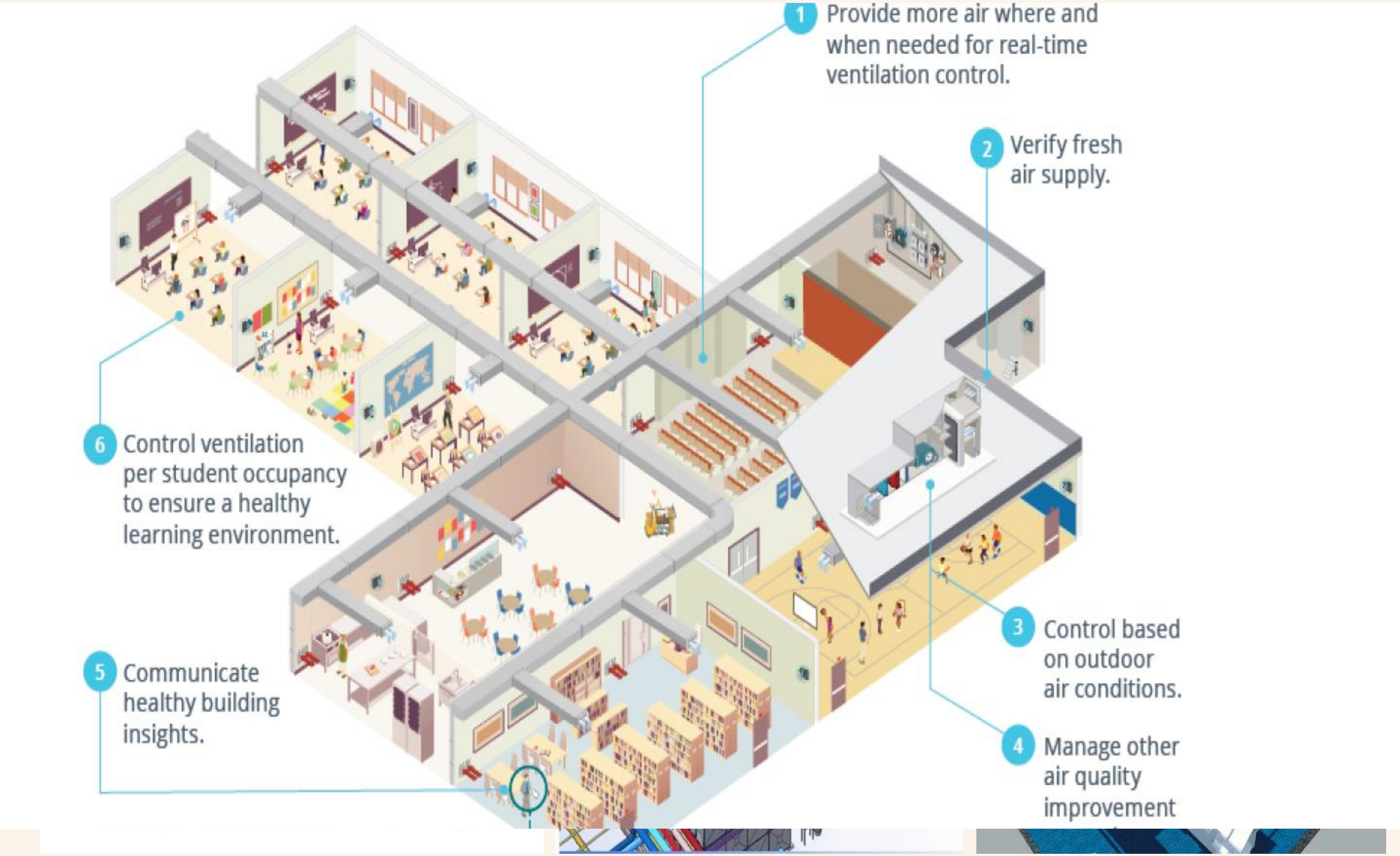
- Using tools like:
  - ✓ Predicted Mean Vote (PMV)
  - ✓ Predicted Percentage Dissatisfied (PPD)
- Implementing HVAC systems to control:
  - ✓ Temperature
  - ✓ Humidity
  - ✓ Air velocity
- Encouraging behavioral adaptations:
  - ✓ Adjustable clothing
  - ✓ Providing personal control over the local environment



Indoor Climate Modelling (ICM)

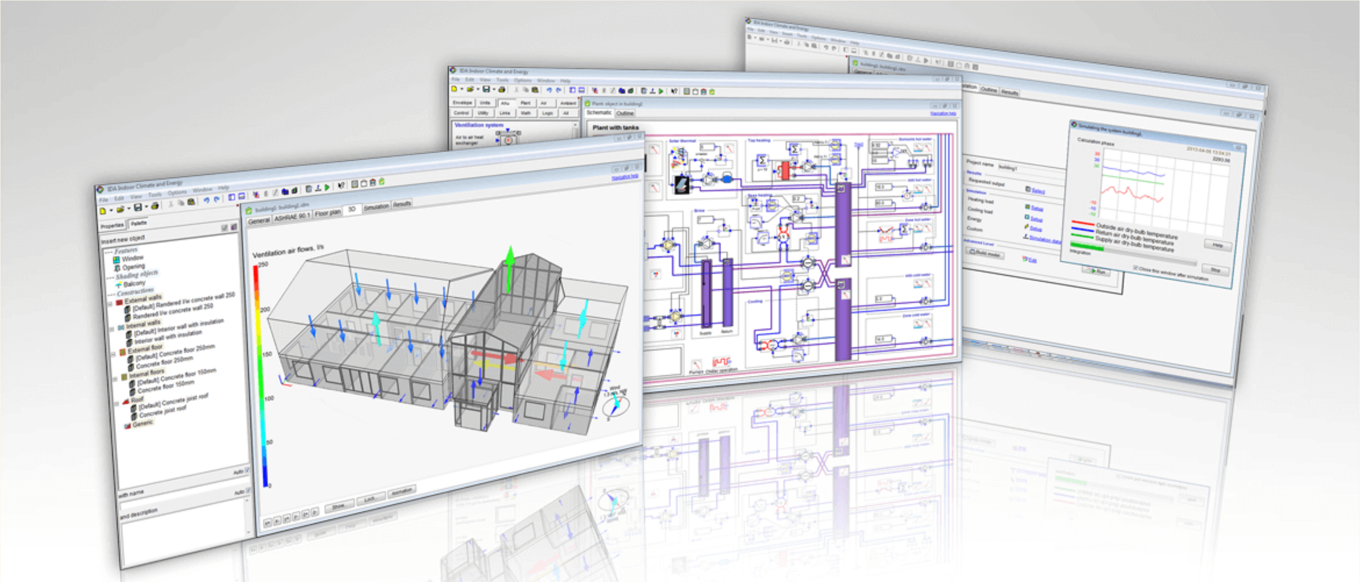
- Indoor climate modelling is the process of using **computer models** to **simulate** the indoor environment.
- It can be used to **predict** the **temperature**, **humidity**, **air quality**, and other factors that affect human comfort and health.
- Indoor climate models are based on the **laws of physics** and **thermodynamics**.
- They take into account the **physical prope**
  well as the activities of the occupants.



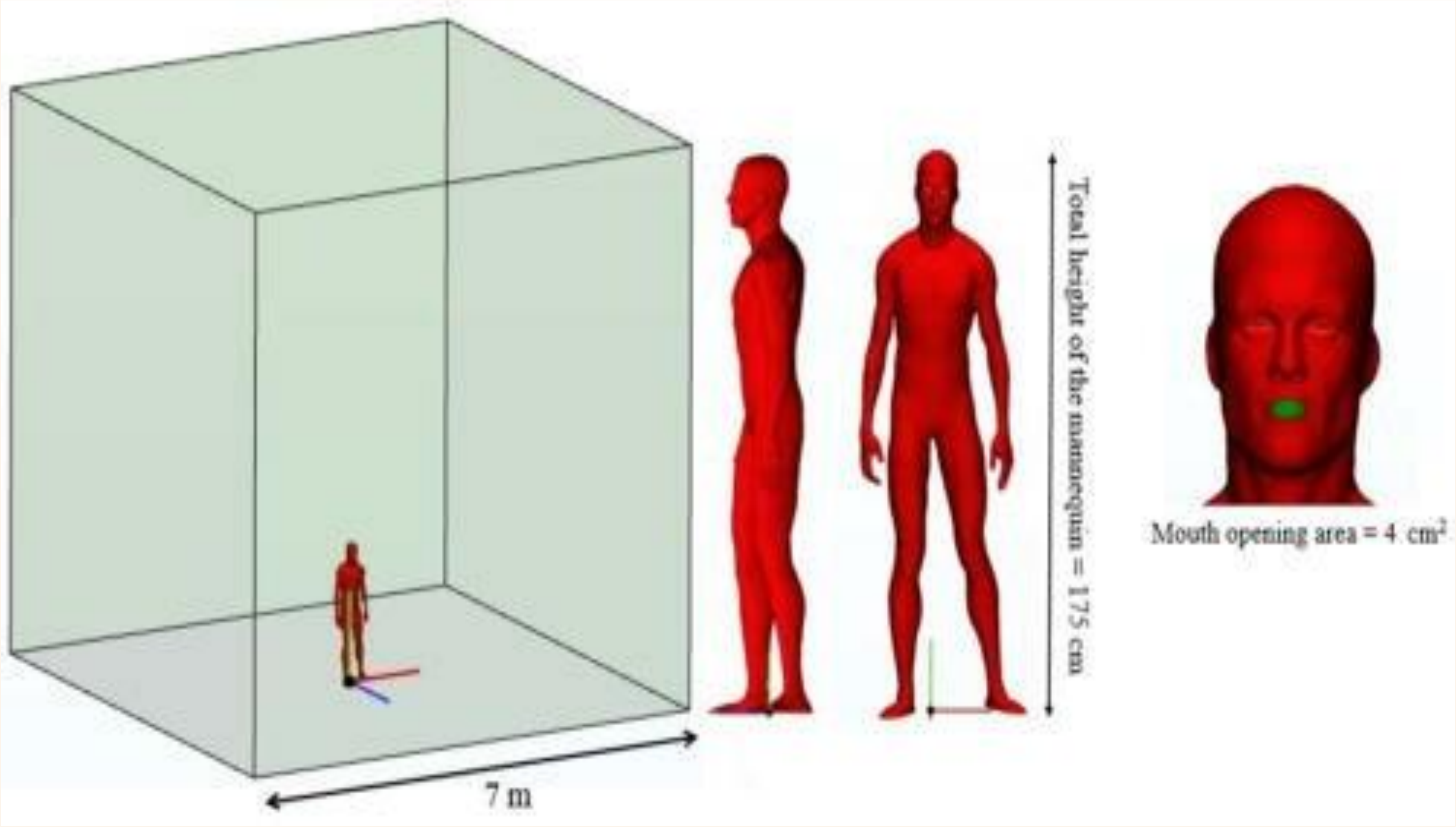

1 Provide more air where and when needed for real-time ventilation control.
2 Verify fresh air supply.
3 Control based on outdoor air conditions.
4 Manage other air quality improvement
5 Communicate healthy building insights.
6 Control ventilation per student occupancy to ensure a healthy learning environment.

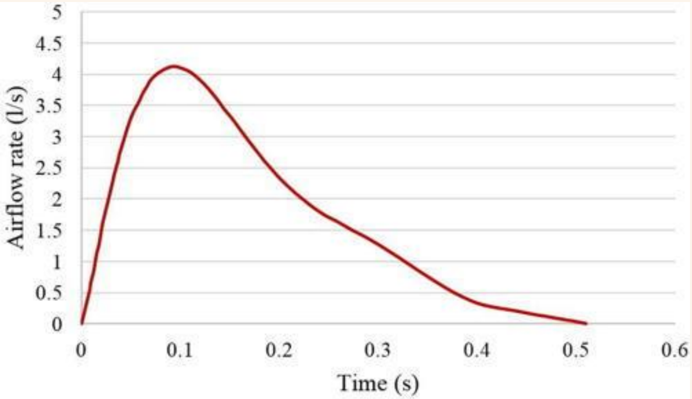
Indoor Climate Modelling

- Stages of Indoor Climate Modelling:
  - **Gathering information** about the indoor environment, **HVAC** systems, and **occupant behavior**.
  - Creating a **computational model** representing the indoor climate.
  - **Running** the model to analyze indoor climate under various conditions.
  - **Data analysis** and **interpretation** to derive actionable insights for improving indoor climate.

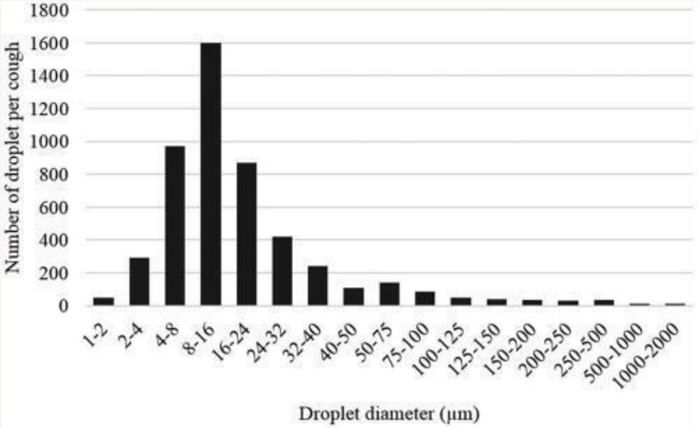


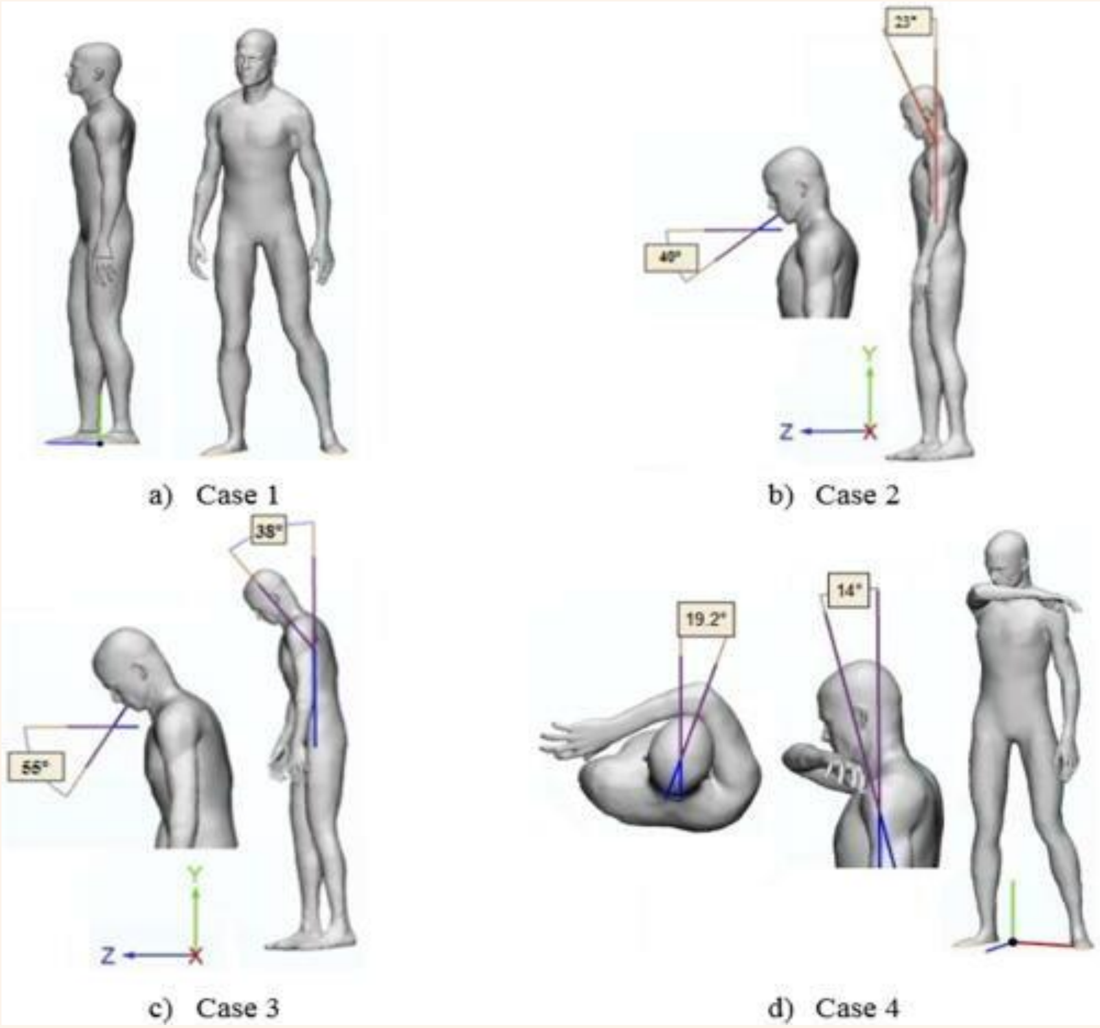
The effect of body position during respiratory activities on the airborne transmission of pathogens





Cough-jet volumetric airflow rate





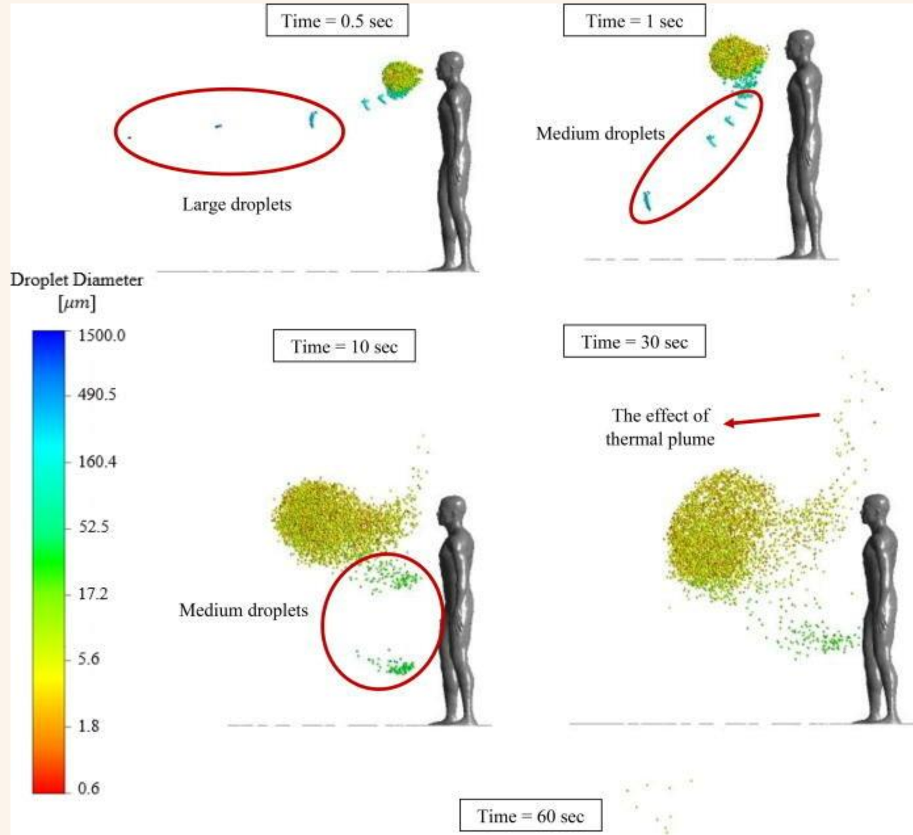
Smaller particles: **>62.5µm**

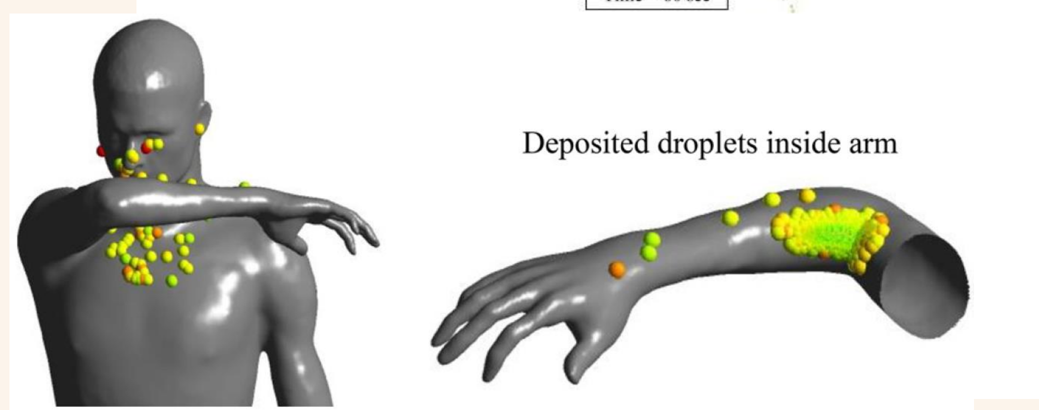
- Travel a **shorter** distance
- Remain airborne **longer**
- Highly affected by thermal plums

Larger particles:

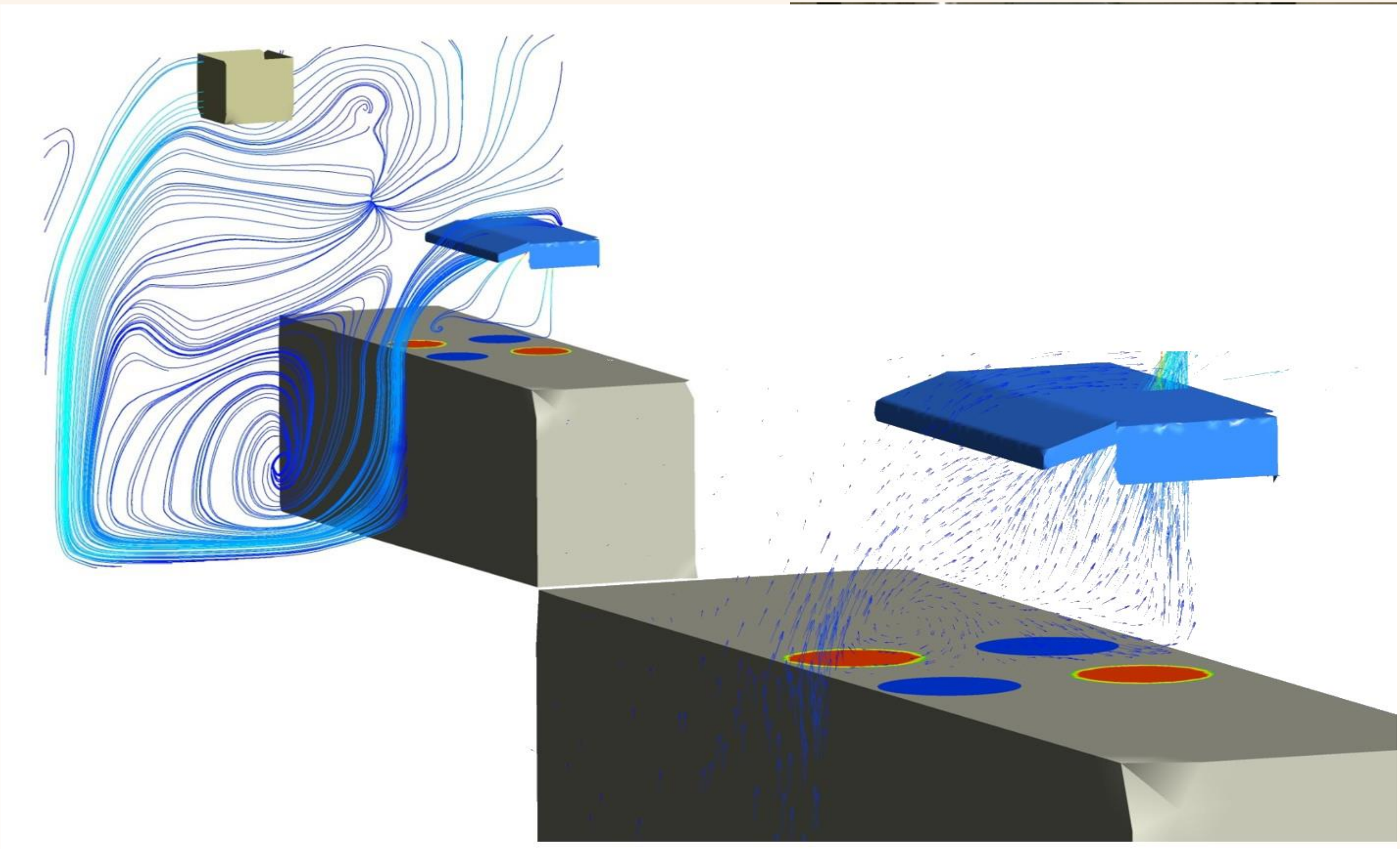
Travel longer in a short time (high initial momentum)

- ➢ **375 µm** on average **0.5m**
- ➢ **750 µm** on average **1.86 m**
- ➢ **1500 µm** on average **2.56 m**



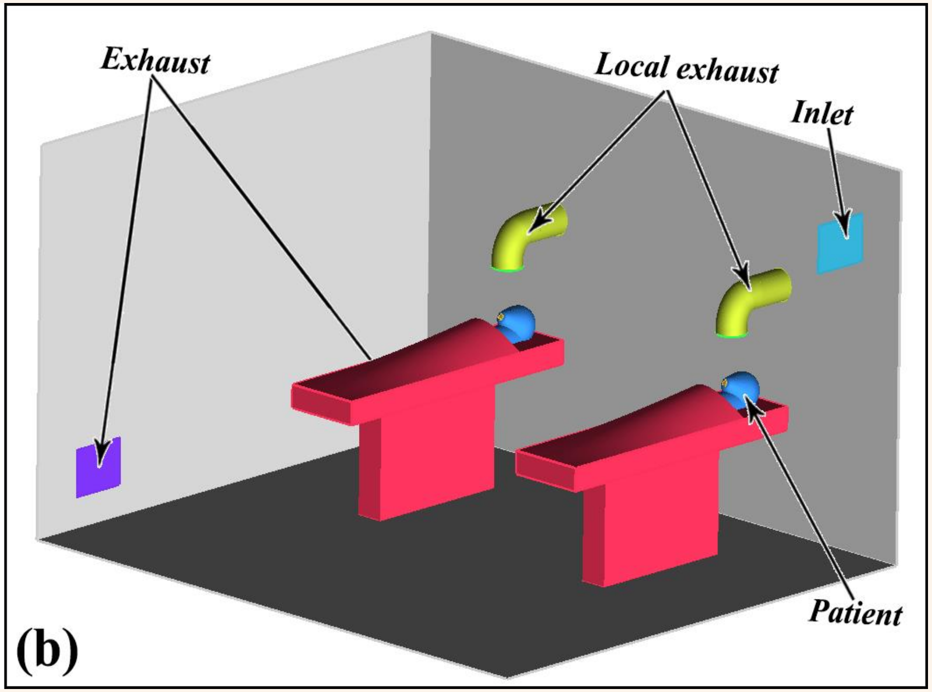


Training and Visualization
How important is it?!!



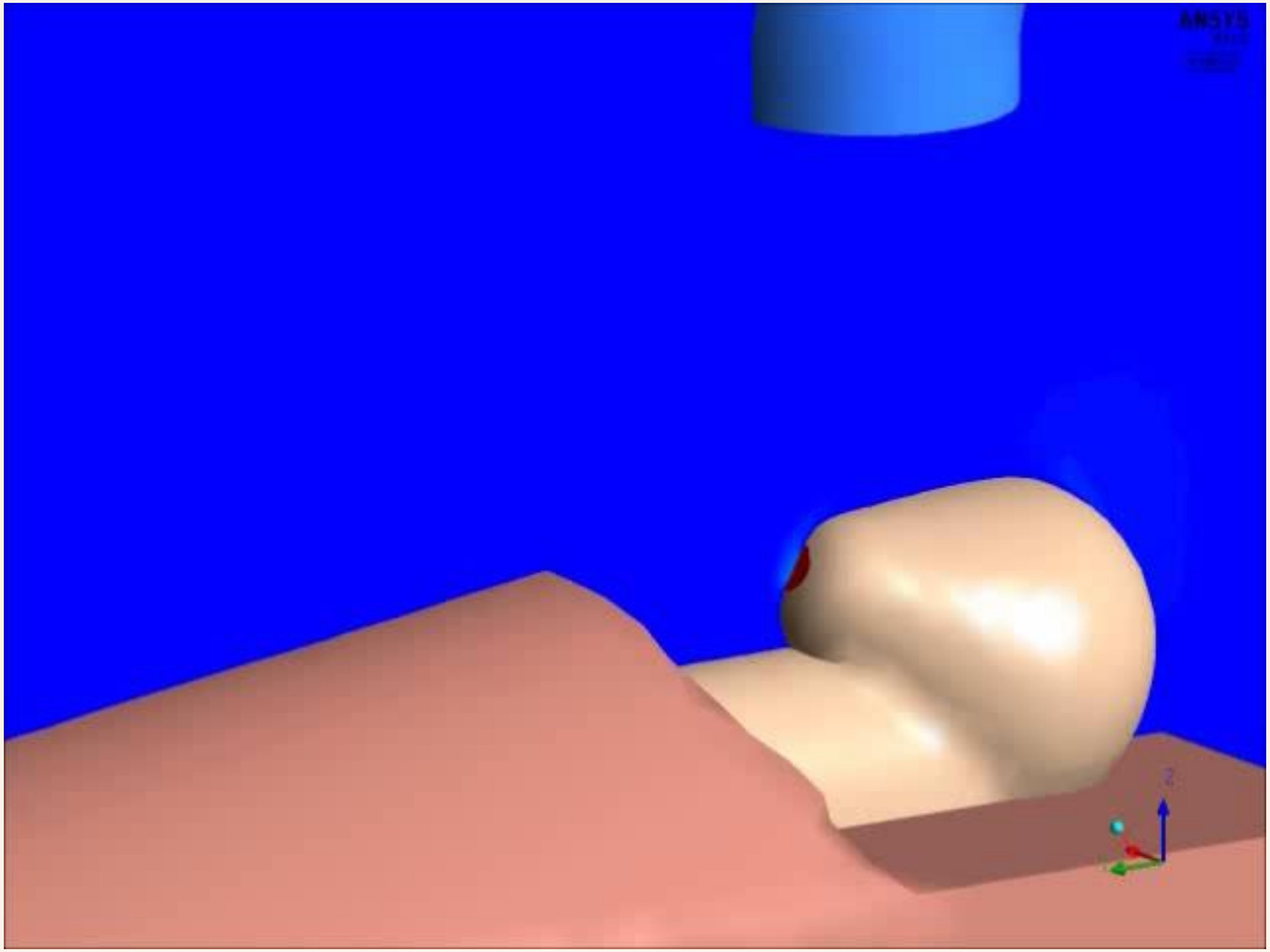
Sadrizadeh, Sasan, et al. "**Measured pollutant removal performance of range-integrated downdraft exhaust kitchen ventilation devices.**" Indoor Air 2018

Individual Exhaust in Multi-bed Hospital Wards

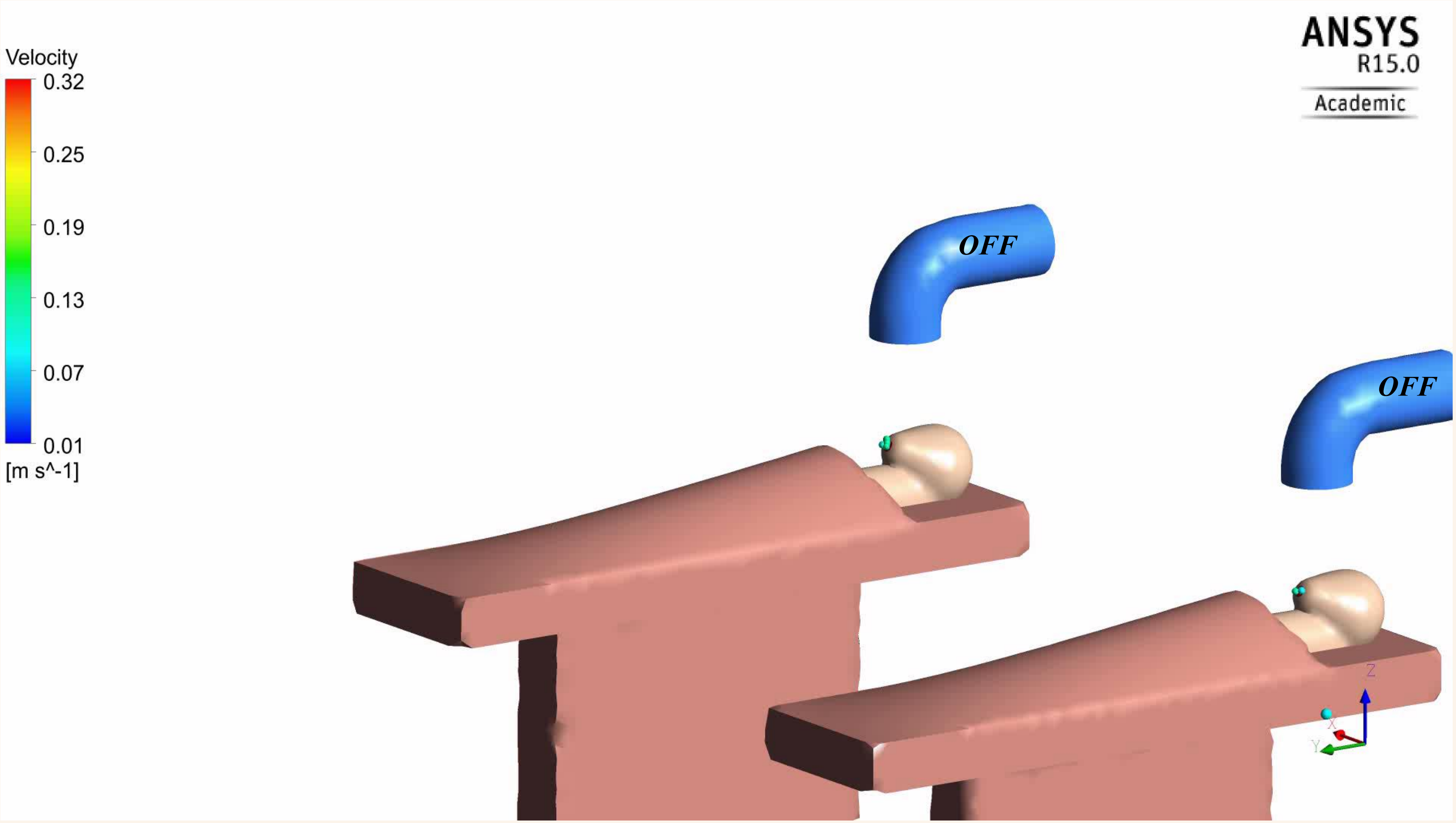


Two-bed hospital ward (a) and its model in CFD simulation (b)

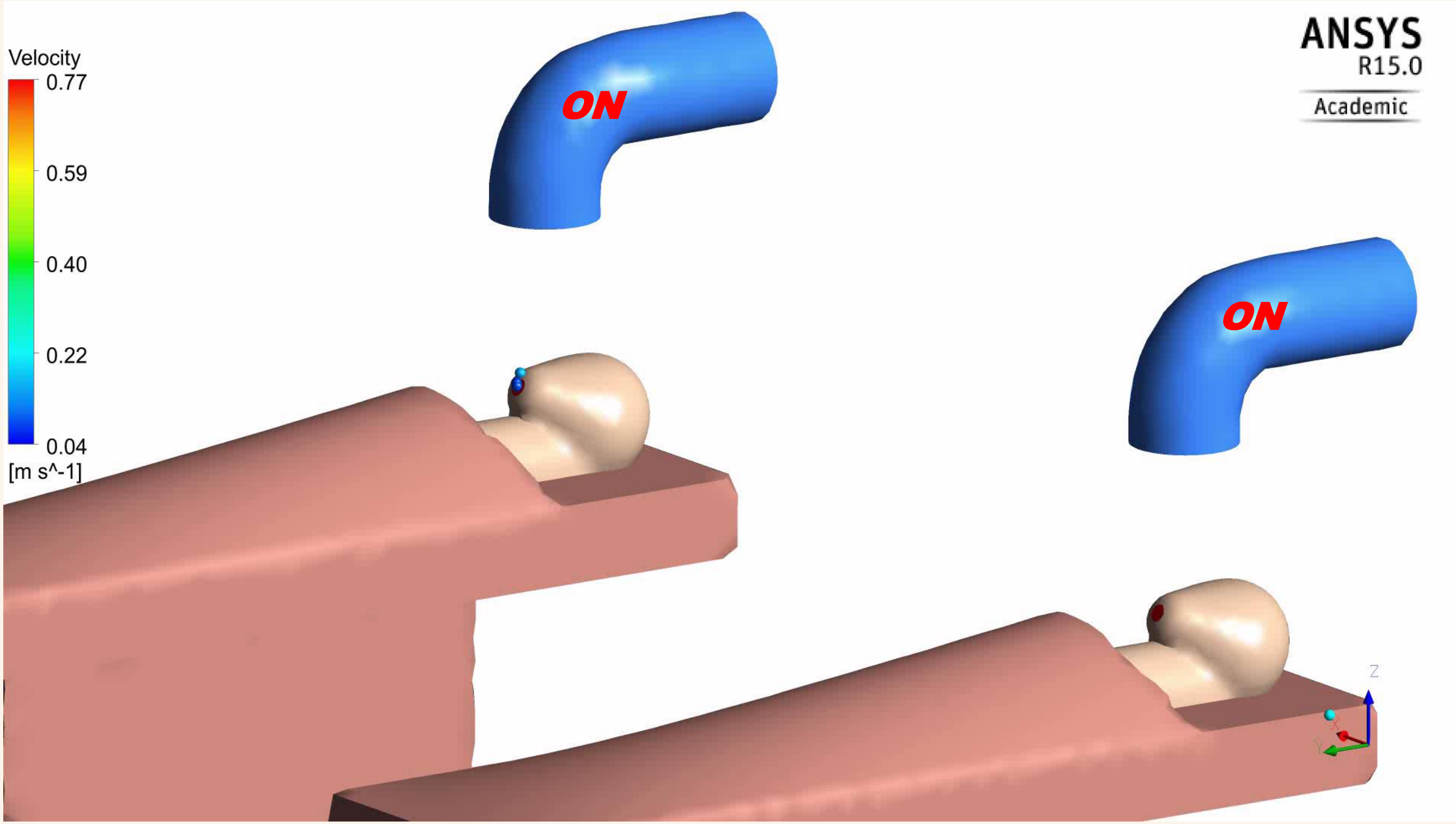
Breading cycle Velocity

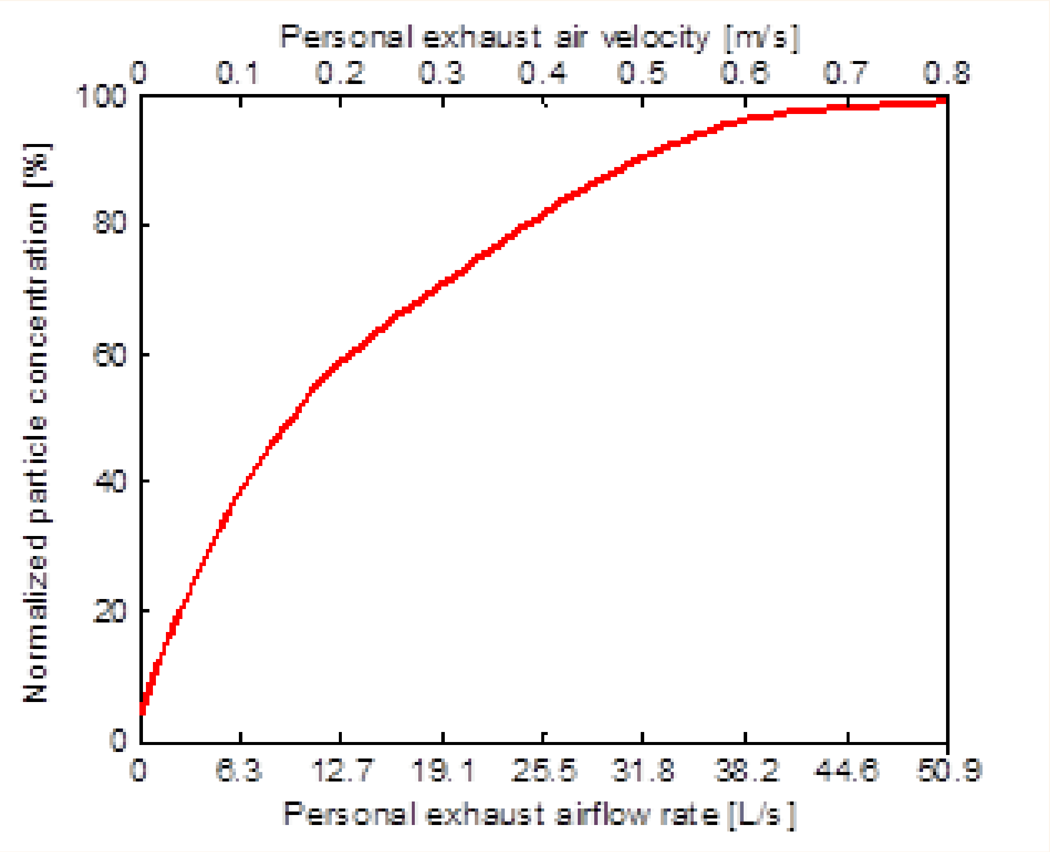


Local Exhaust fan OFF

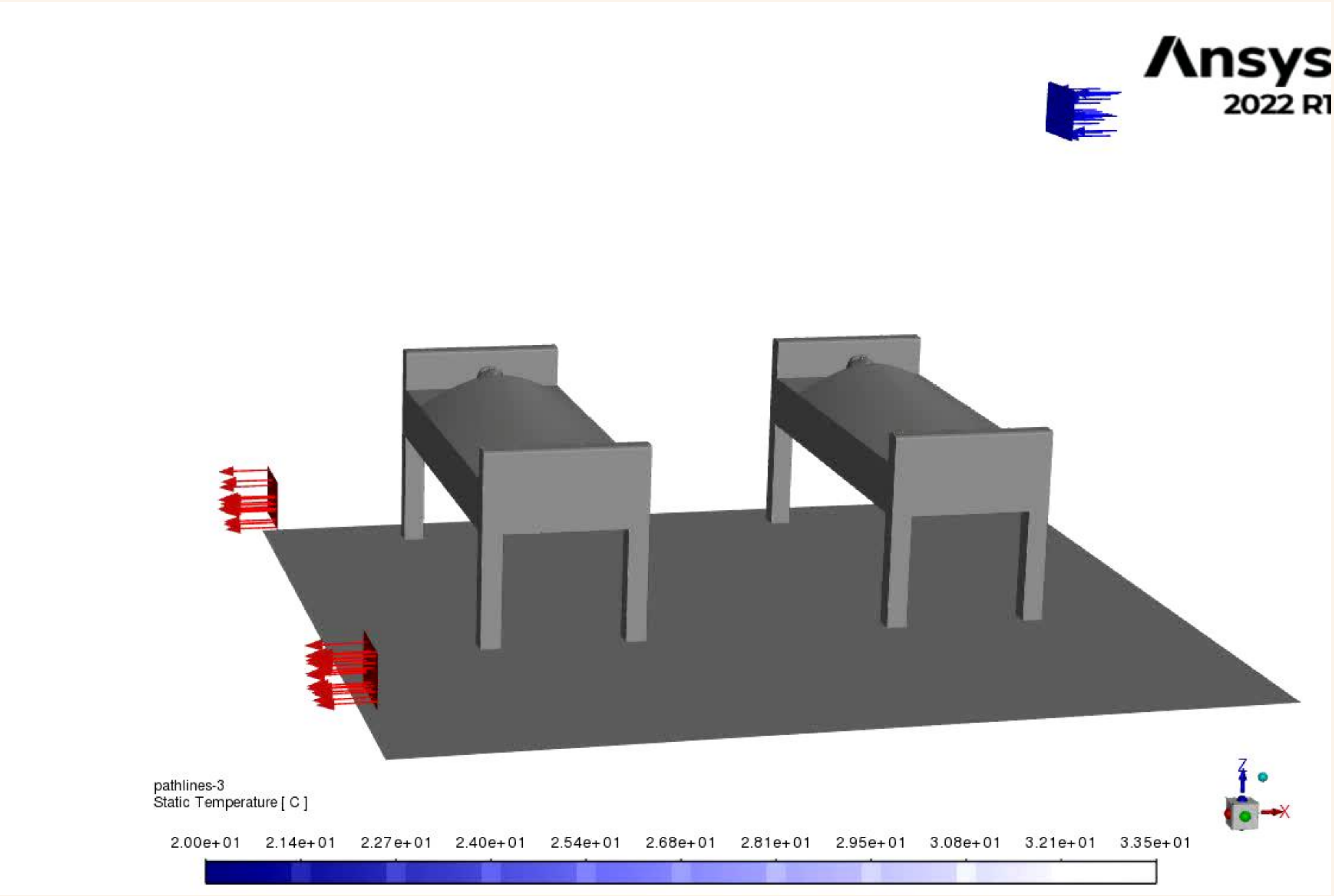


Local Exhaust fan ON





Percentage concentration of water droplets as a function of individual exhaust airflow rate; particle concentration was normalized by the patient's mouth concentration


Ansys
2022 R1
pathlines-3
Static Temperature [C]
2.00e+01
2.14e+01
2.27e+01
2.40e+01
2.54e+01
2.68e+01
2.81e+01
2.95e+01
3.08e+01
3.21e+01
3.35e+01
Z
X

Influence of staff number and internal constellation on SSI

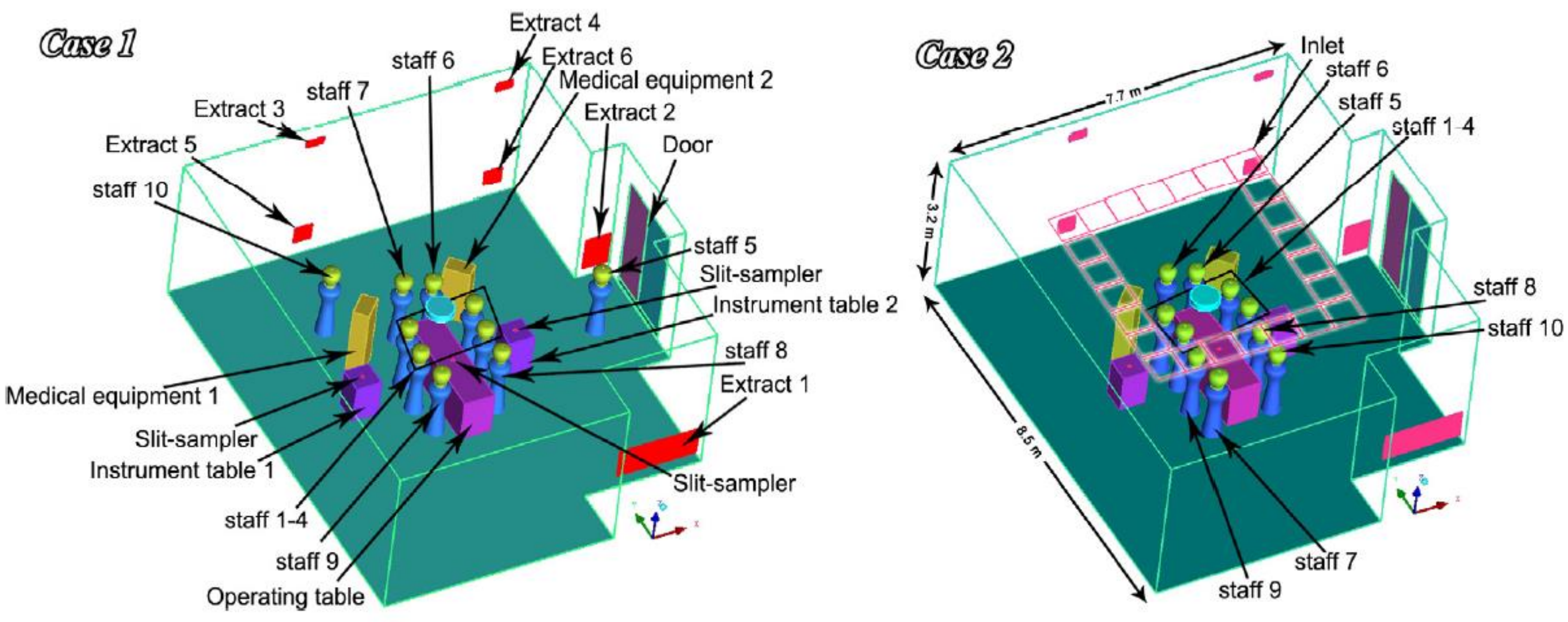


Fig. 1. Isometric view of operating room model: 1st staff configuration (left), 2nd staff configuration (right).

Influence of staff number and internal constellation on SSI

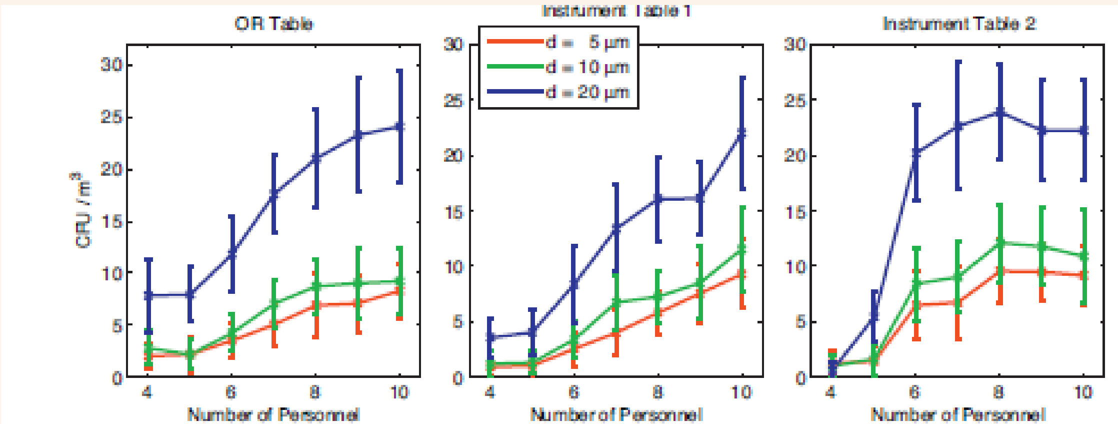
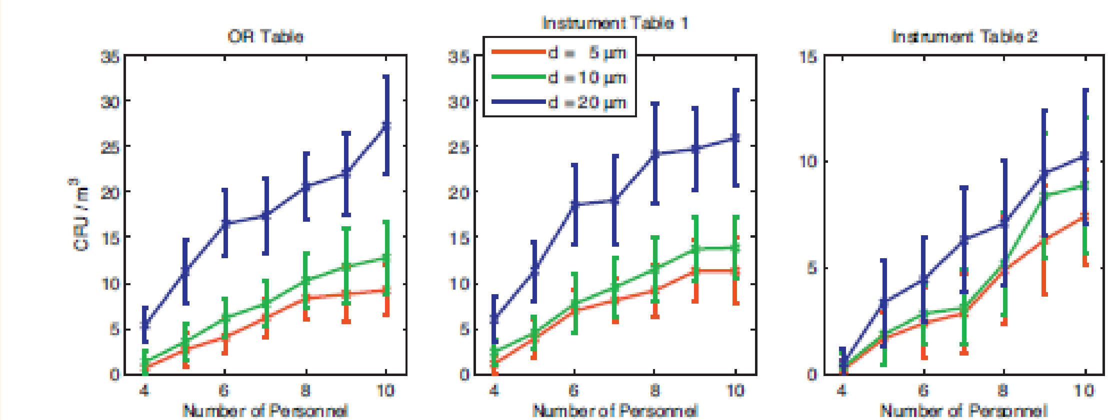


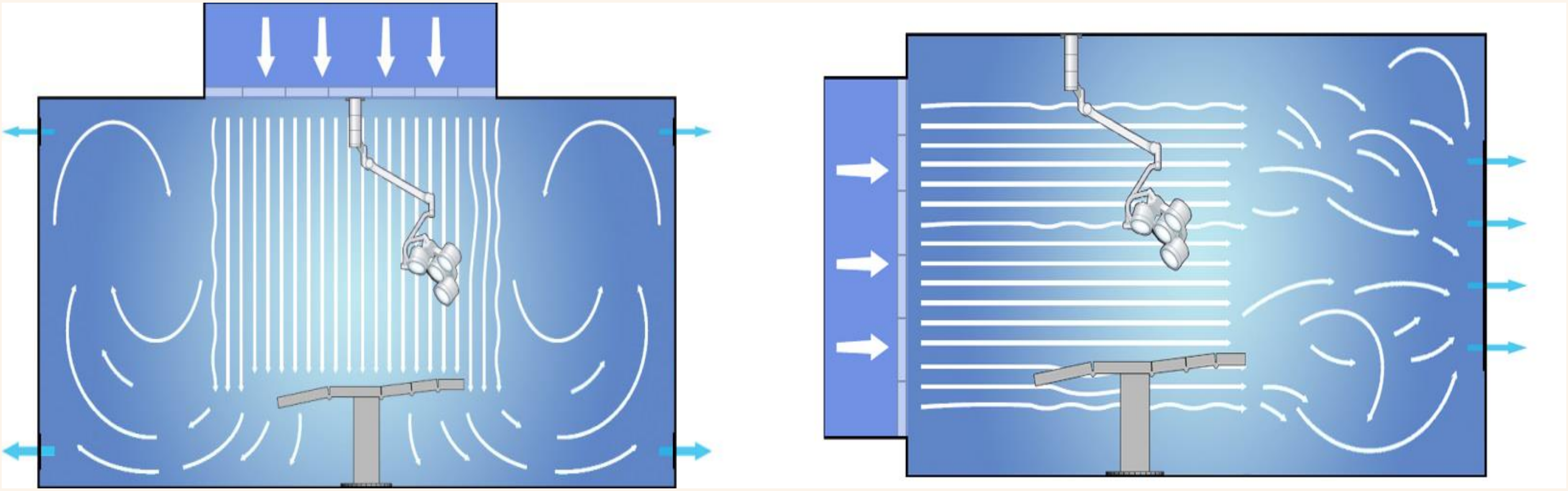
Fig. 6. The simulated BCP concentrations as a function of staff number at different particle diameters for active sampling method and staff configuration Case 1.



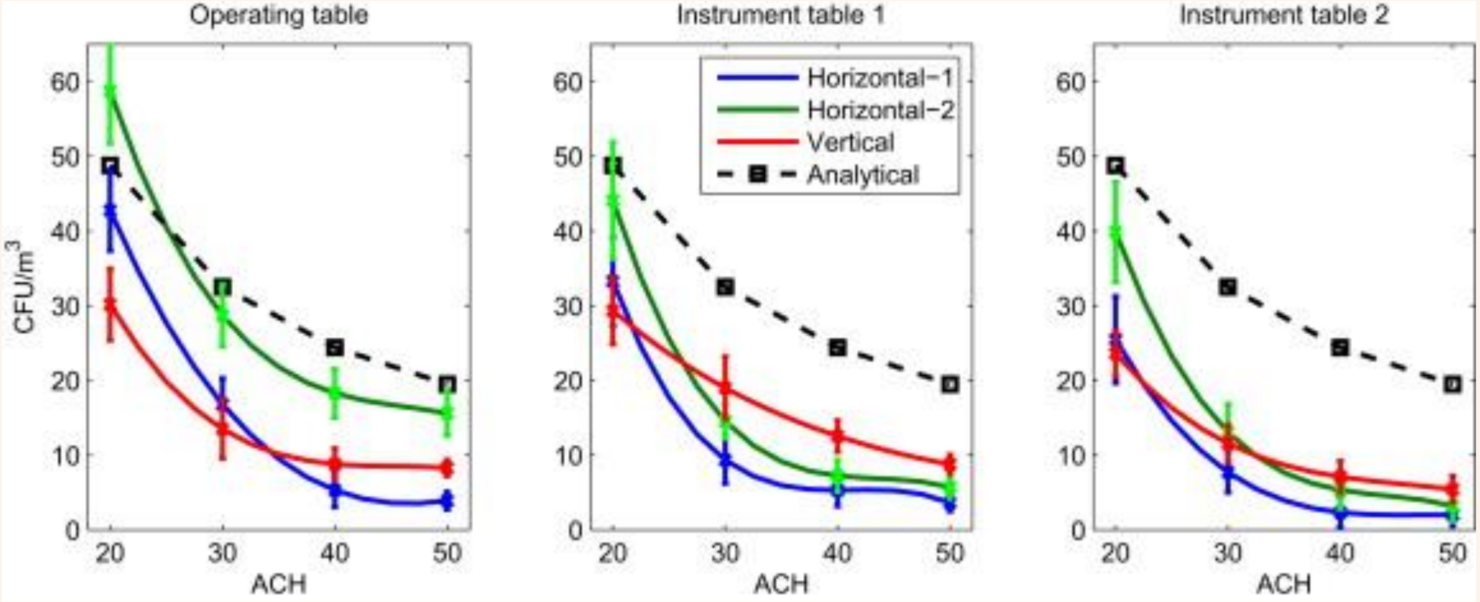
Airborne particle concentration versus number of surgical staff in the OR

The more person in the OR, the higher the contaminant concentration

Comparison between LAF ventilation systems

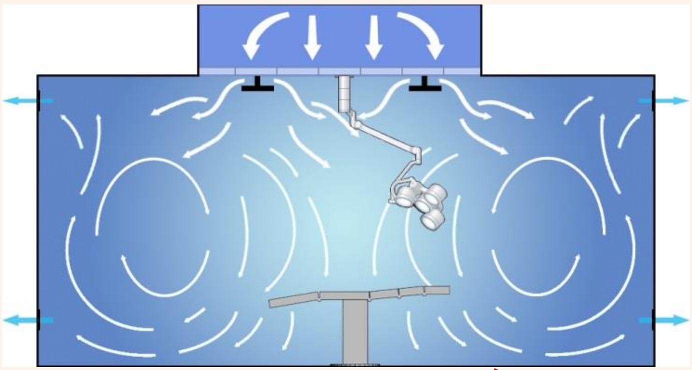


Comparison between LAF ventilation systems

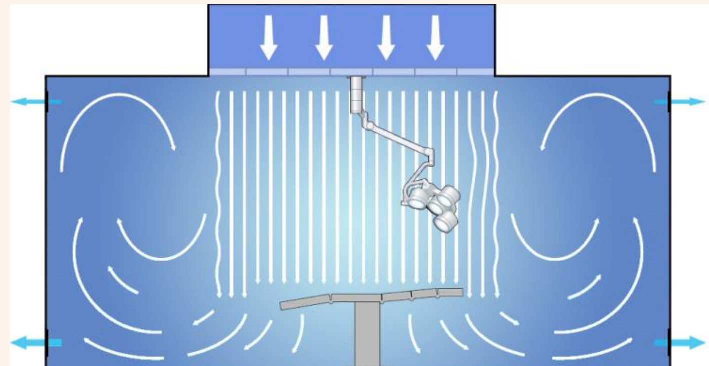


Airborne particle concentration for both the ventricle and the horizontal ventilation system. An analytical calculation was also presented as a reference.

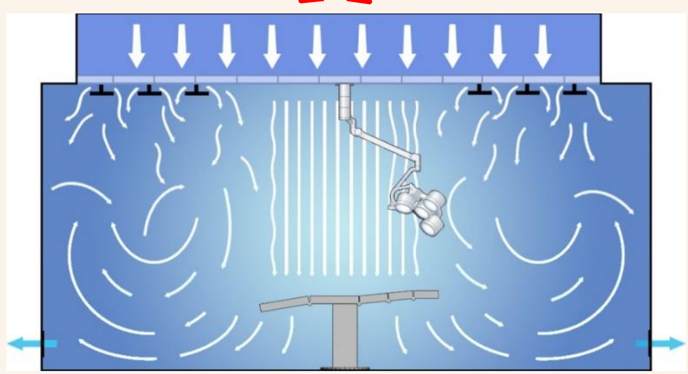
Hybrid Ventilation



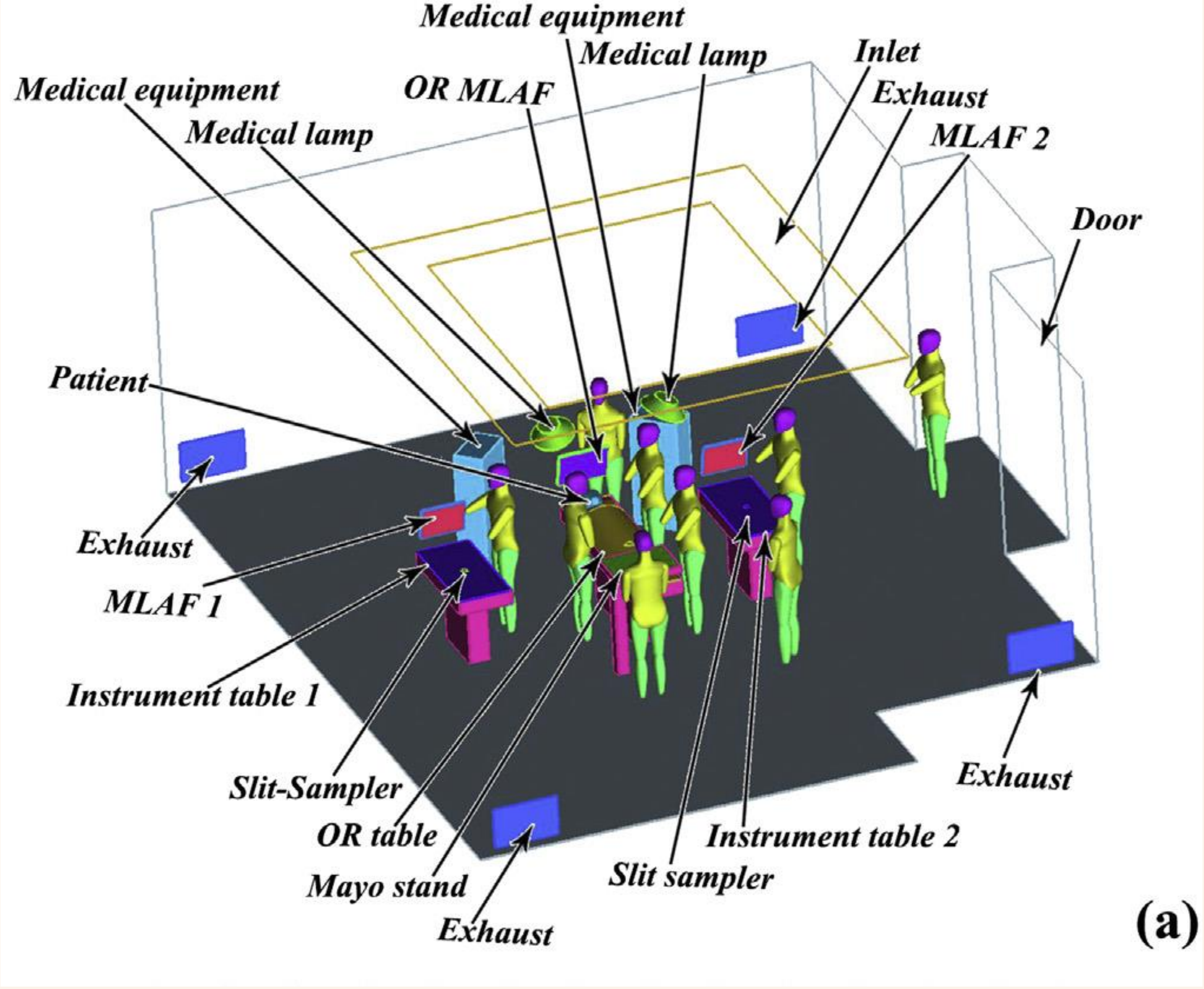
Mixing



LAF

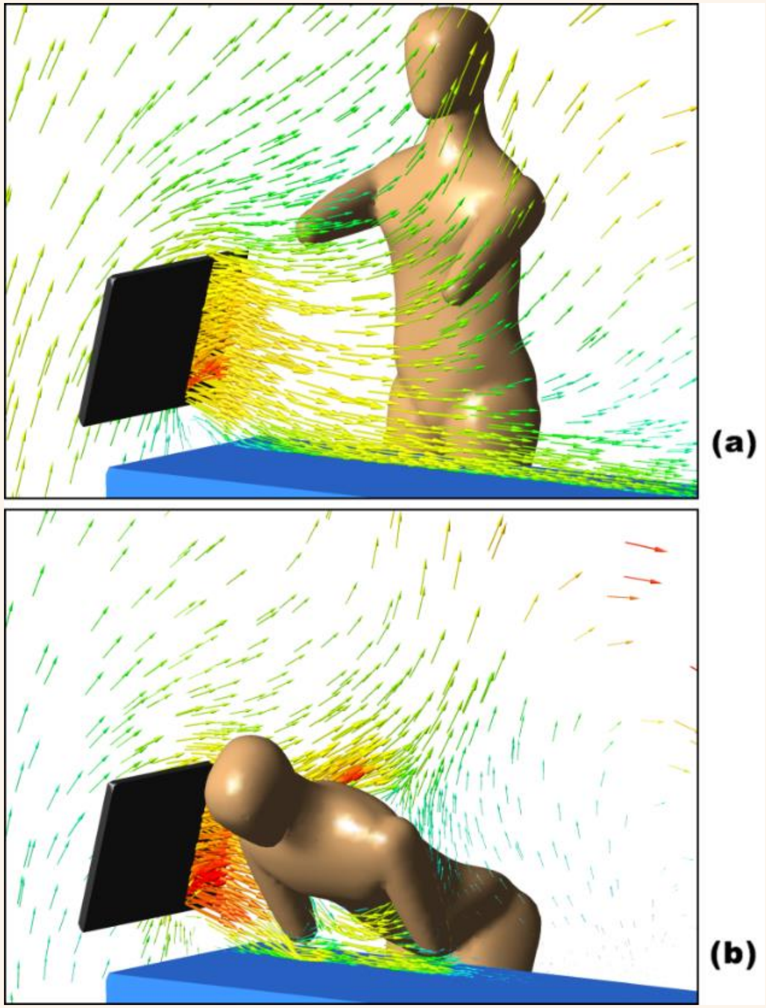


Impact of surgeon posture on airborne particle distribution

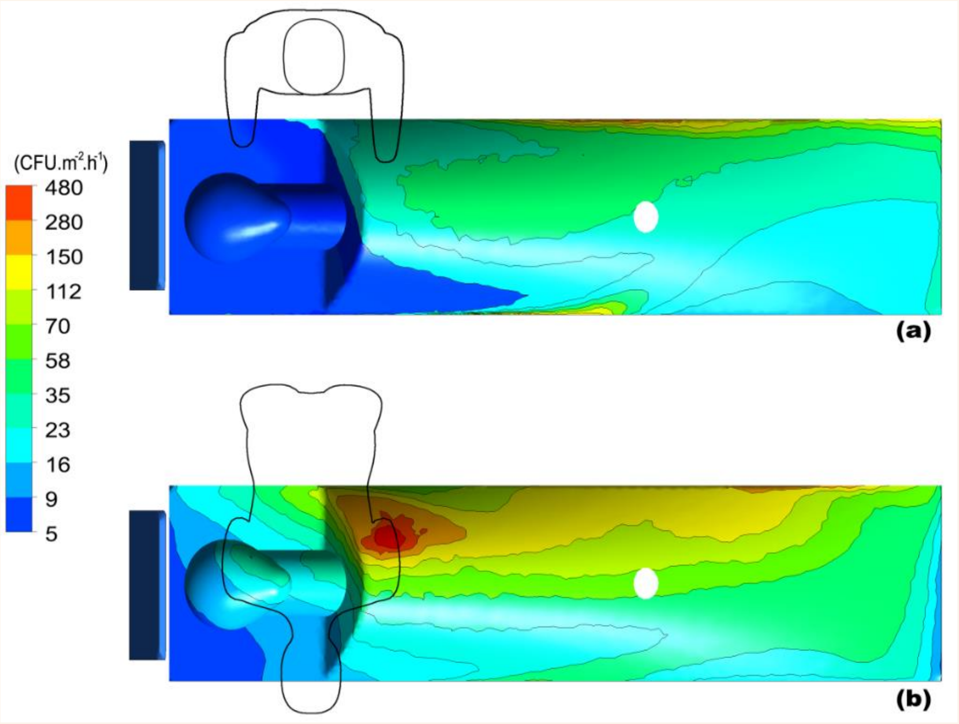


Local portable laminar airflow ventilation

Mobile exponential ultra-clean air units (MLAFs).
Inlet velocity considered in the range of 0 to 1 m/s

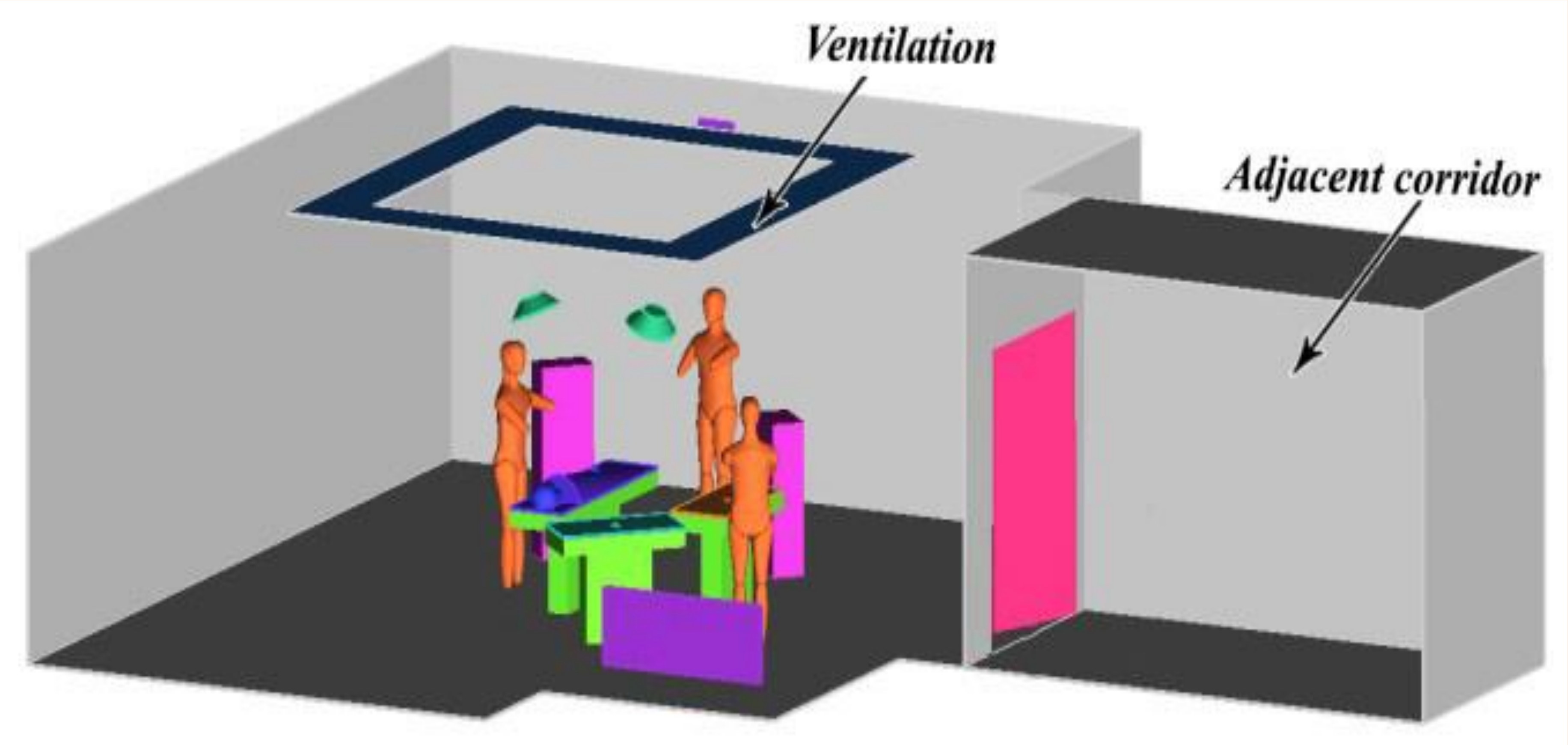


Velocity vector plot at the centerline plane of ultraclean-zoned ventilation above the instrument table



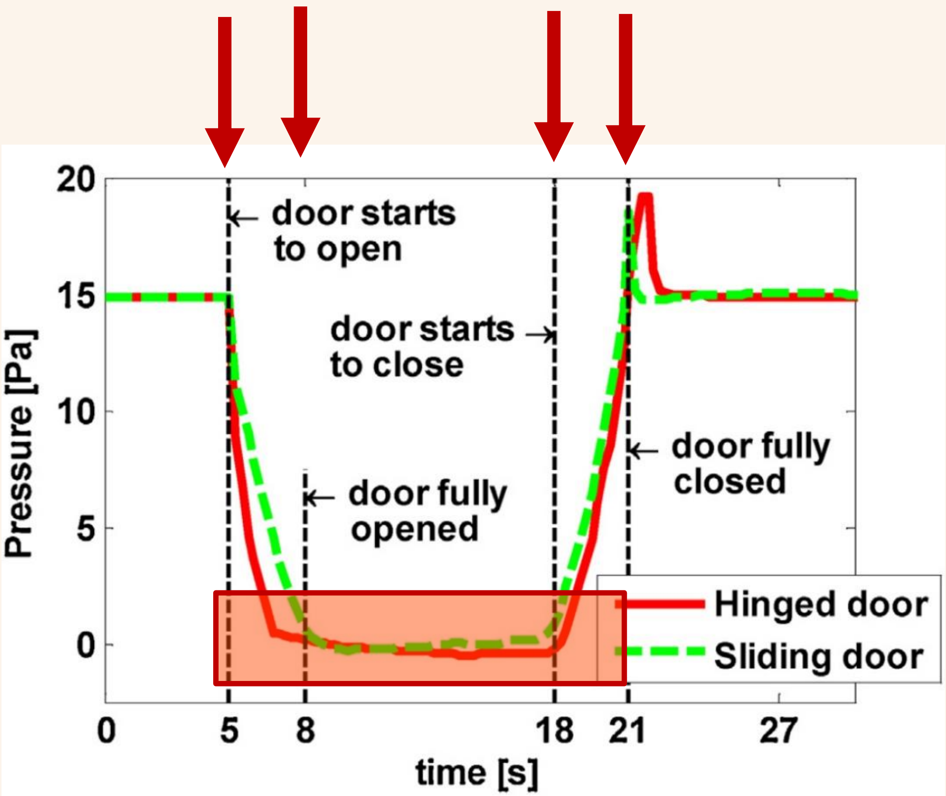
Bending posture blocks the airflow field from the ceiling diffusers. This results in the accumulation of infectious particles in the surgical area.

Hinged and Sliding Door Opening in Operating rooms

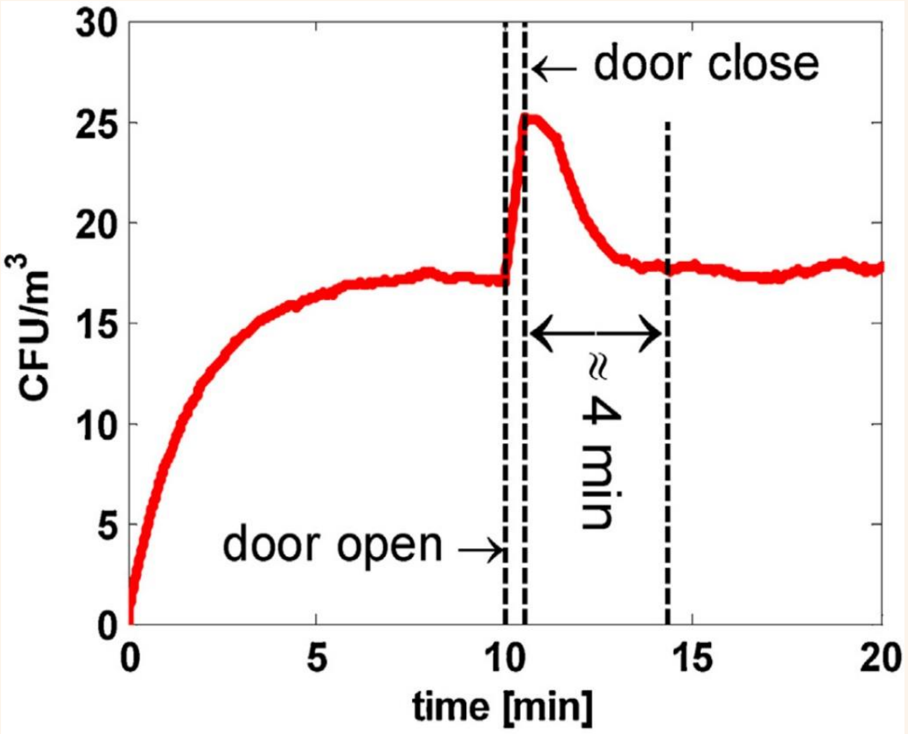


An OR with airflow rate of 2.5 m^3/s results in ACH = 70 h^{-1}

Door Opening: Pressure and Contaminant Concentration

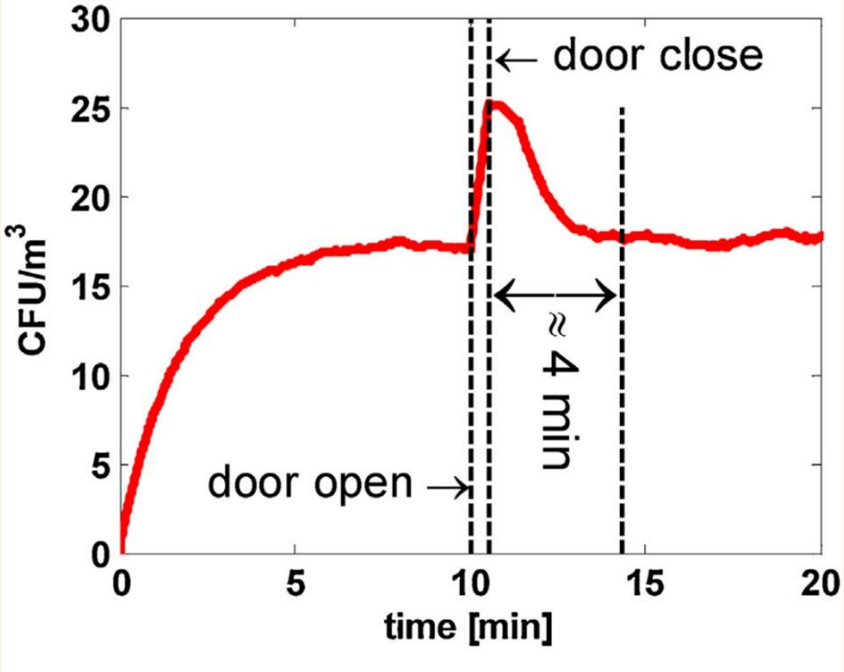


Pressure variation as a function of time during a door opening cycle (in steady state situation).

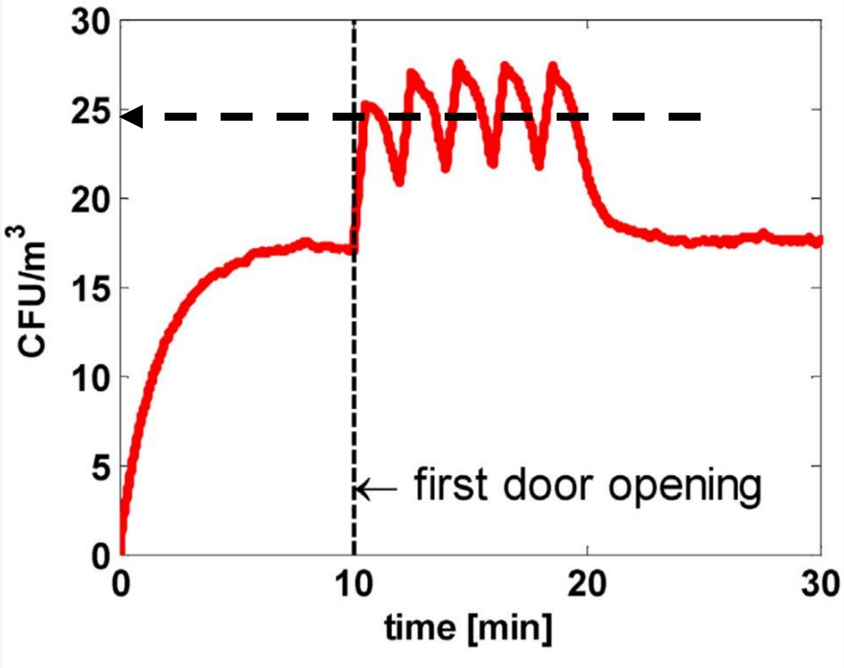


OR recovery time in a single cycle of door-opening.

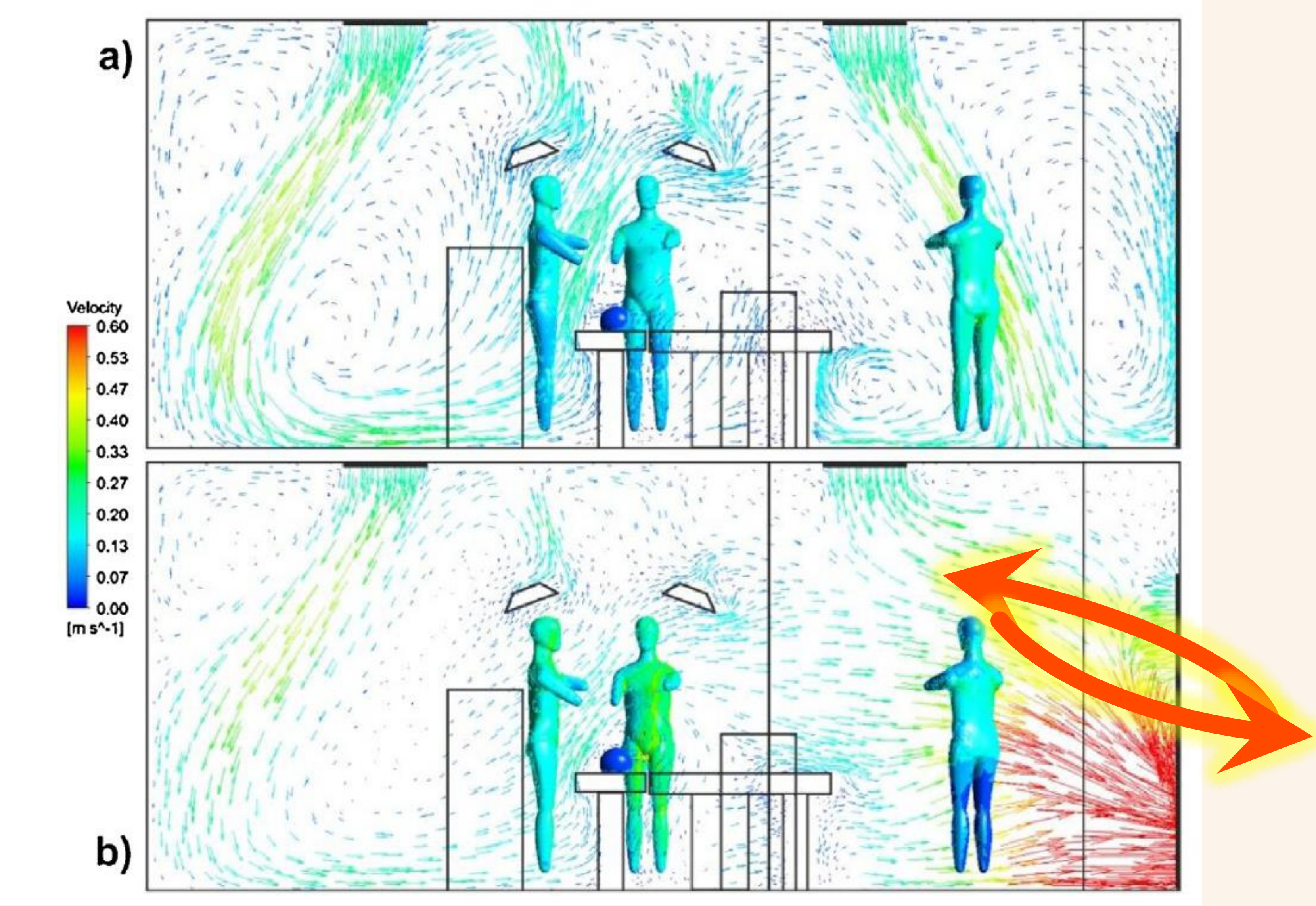
Single and Multiple Door Openings



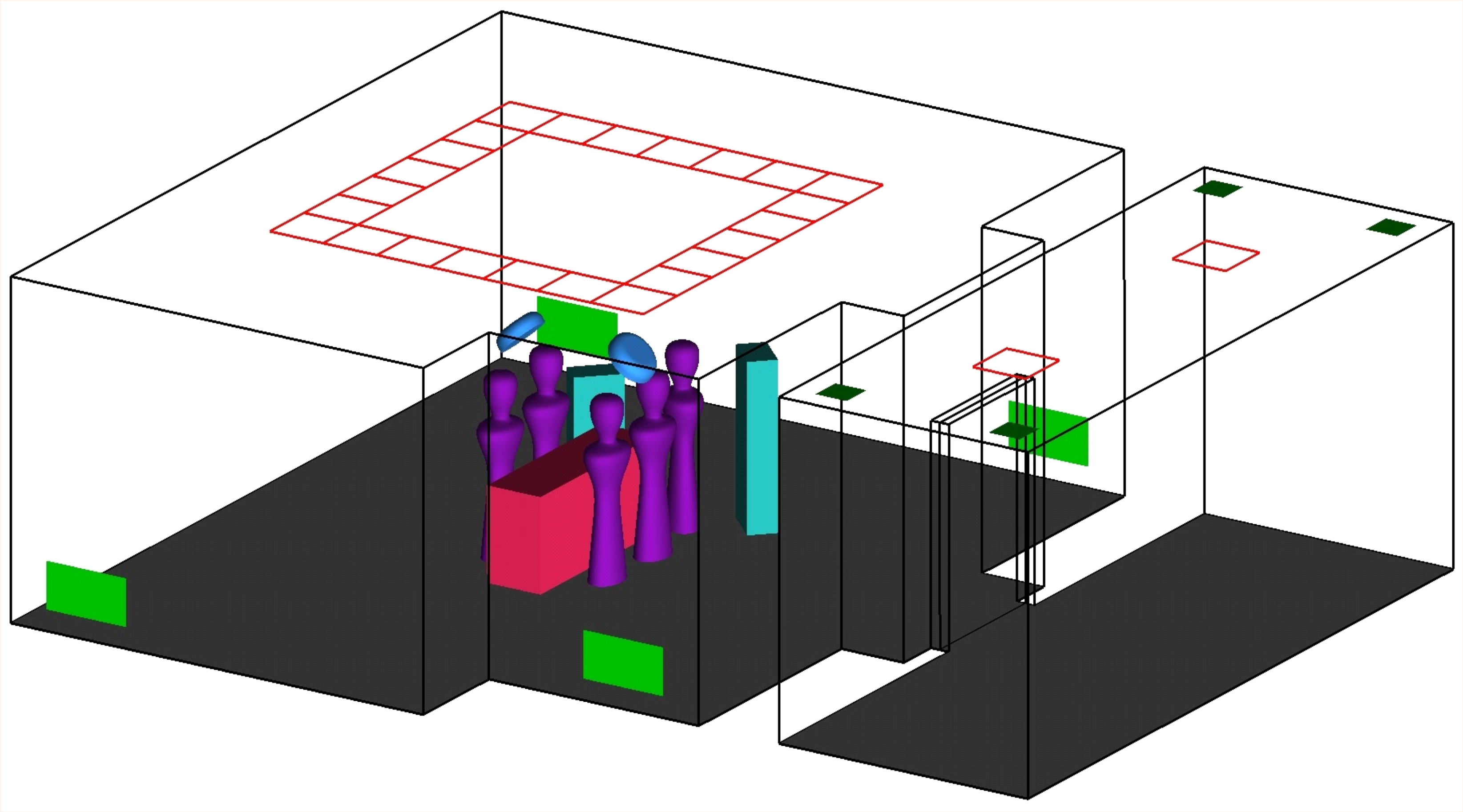
OR recovery time in a single cycle of door-opening.



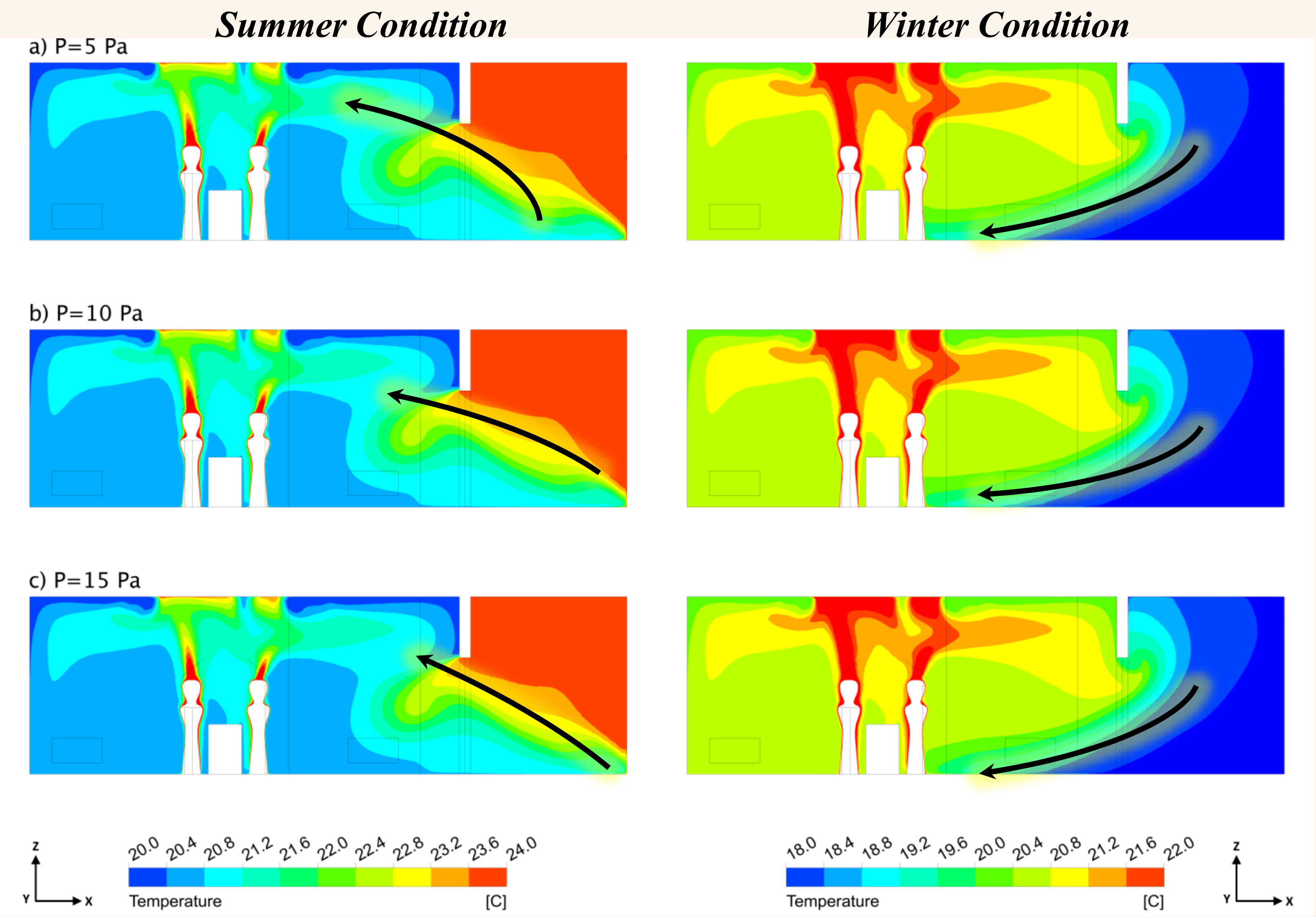
OR recovery time in five cycles door-opening every two and half minutes



Airflow vector plots on a plane passed through the door (a) door close; (b) door open.



Isometric view of the operating room and the adjacent corridor.

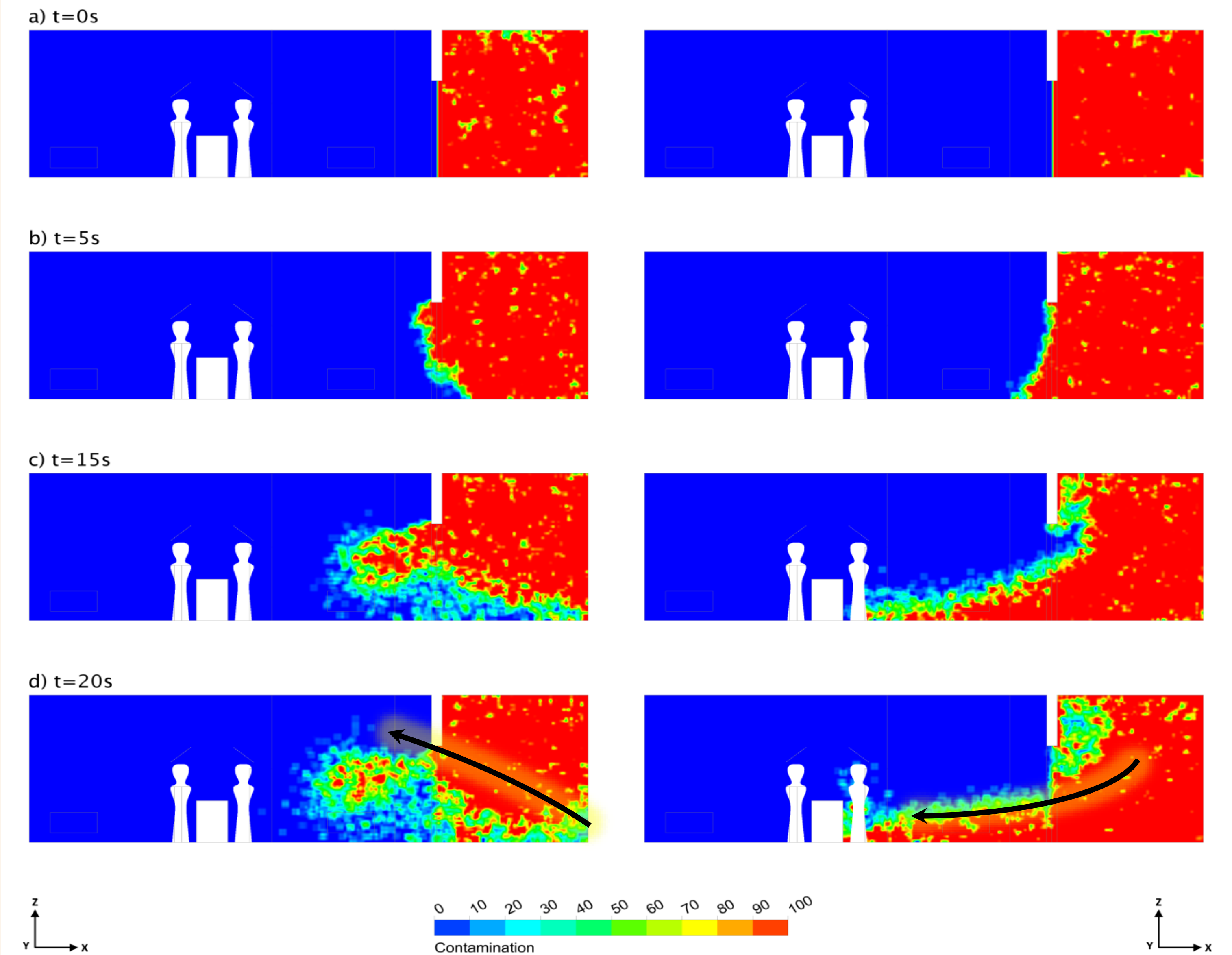


Temperature contour plot at the center plane of the sliding door at t=15s

Summer Condition: Corridor air has a higher temperature than the OR

Winter Condition: Corridor air has a lower temperature than the OR

Summer Condition | *Winter Condition*



Airborne BCPs concentration at the center-plane of the sliding door

Adjust the outflow rate during door opening and passage

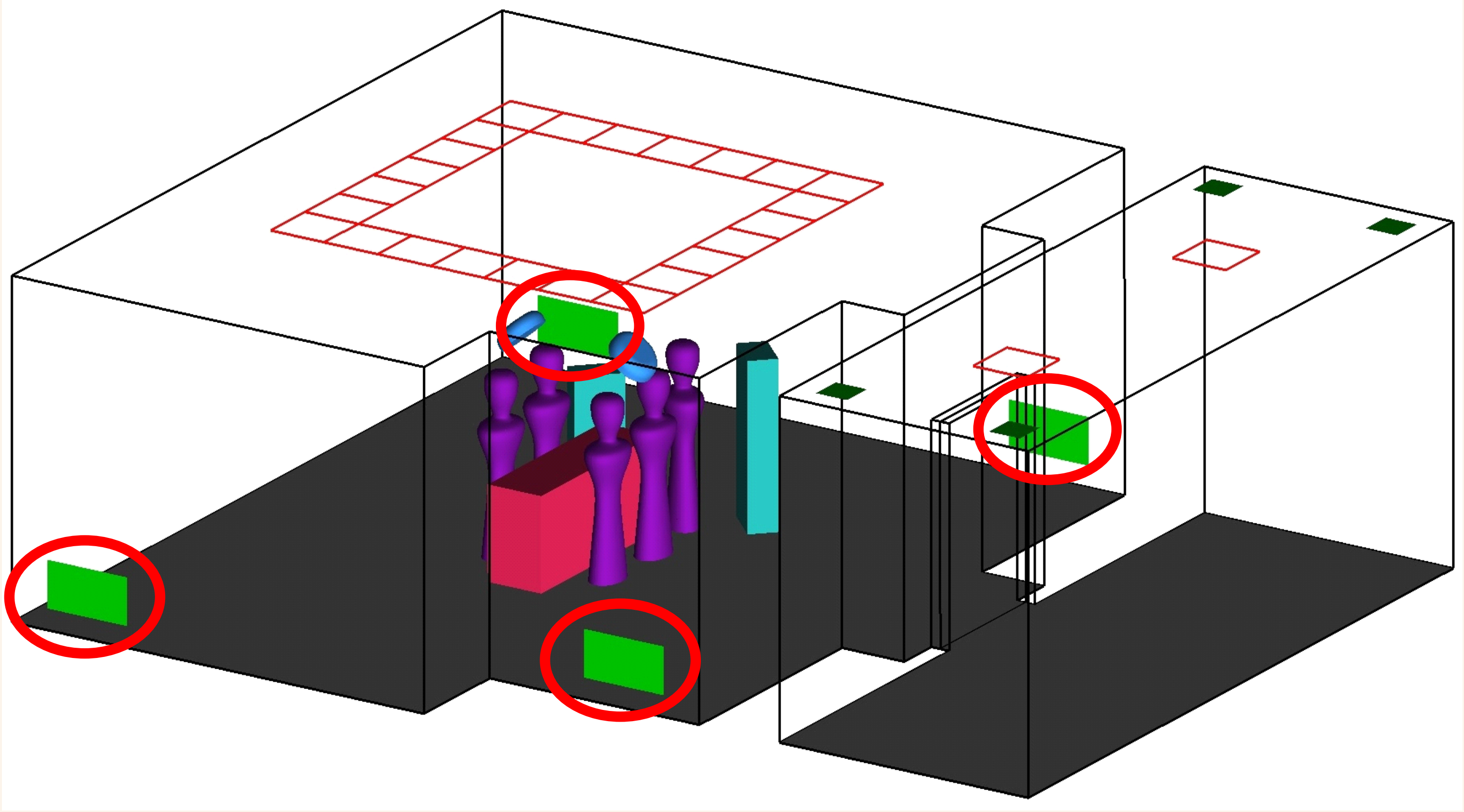
Reduce the outlet airflow rate by

Case A: 0 %

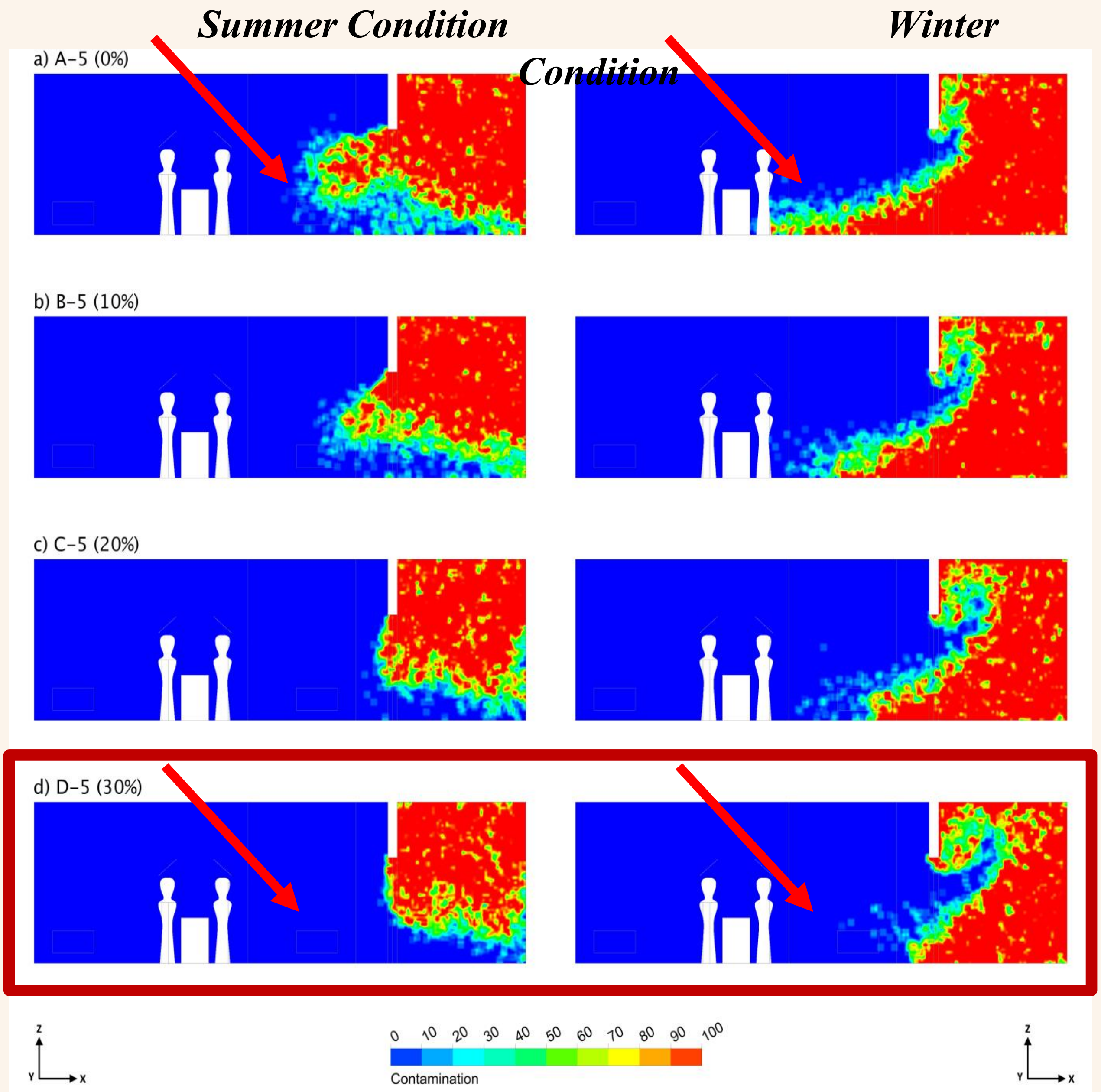
Case B: 10 %

Case C: 20 %

Case A: 30 %



Airborne particle concentration at the center-plane of the sliding door at (t = 15s)



Outlet airflow rate reduced by:

a) 0%

b) 10%

c) 20%

d) 30%

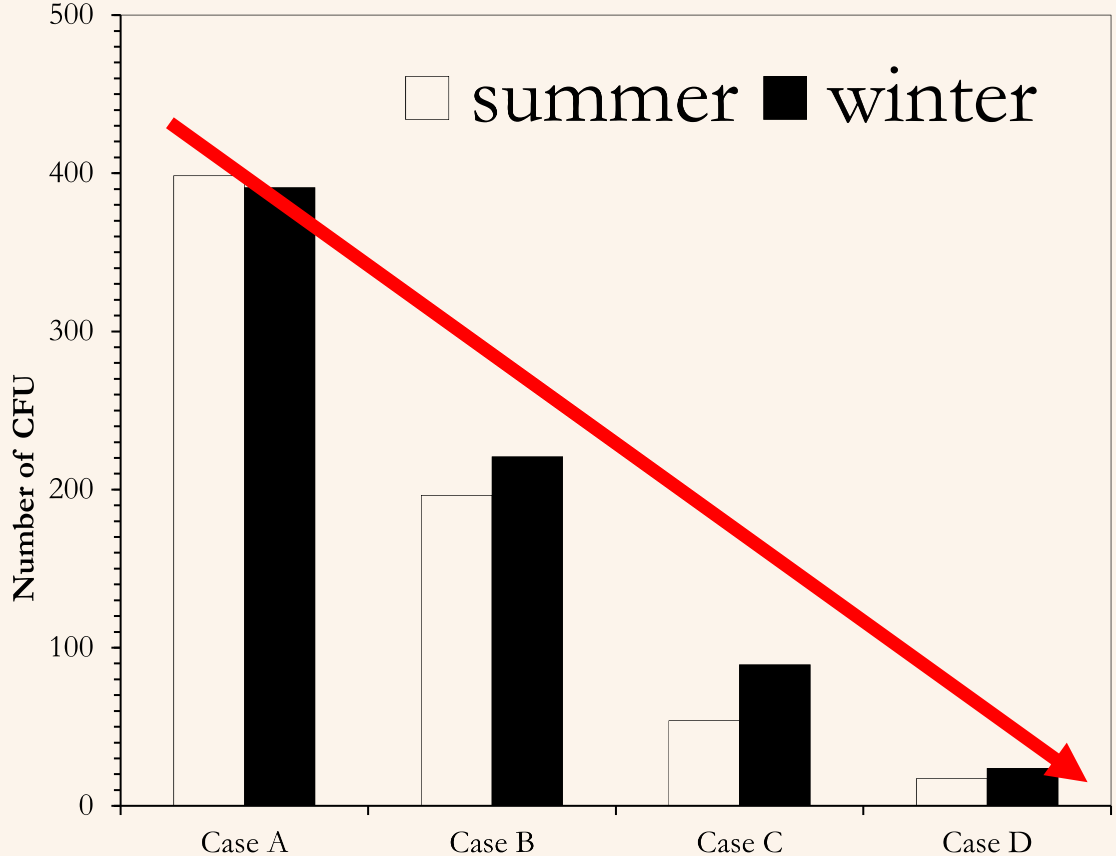
*Outlet airflow rate **reduced** by:*

Case A: 0%

Case B: 10%

Case C: 20%

Case D: 30%



Cong W et al 2019

What if we increase the airflow rate through the outlets in case of Negative isolation rooms?

Predicted Mean Vote

Predicted Percentage of Dissatisfied

Draft Rate