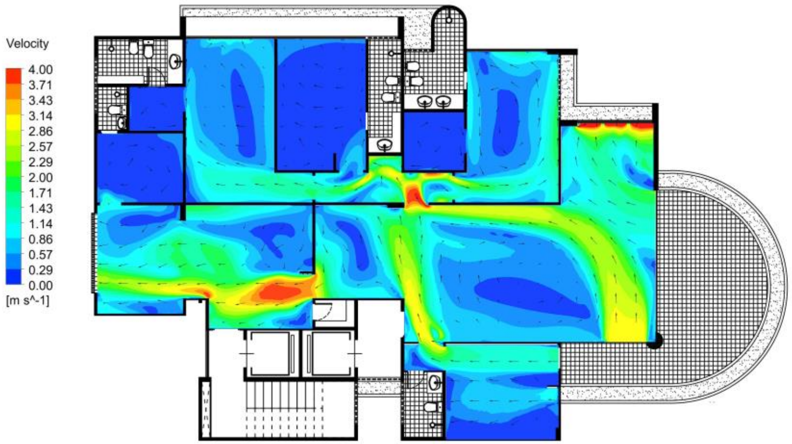
LISC

LABORATORY FOR INTELLIGENT SYSTEMS AND CONTROLS

Indoor Airflow Modeling and Data Assimilation

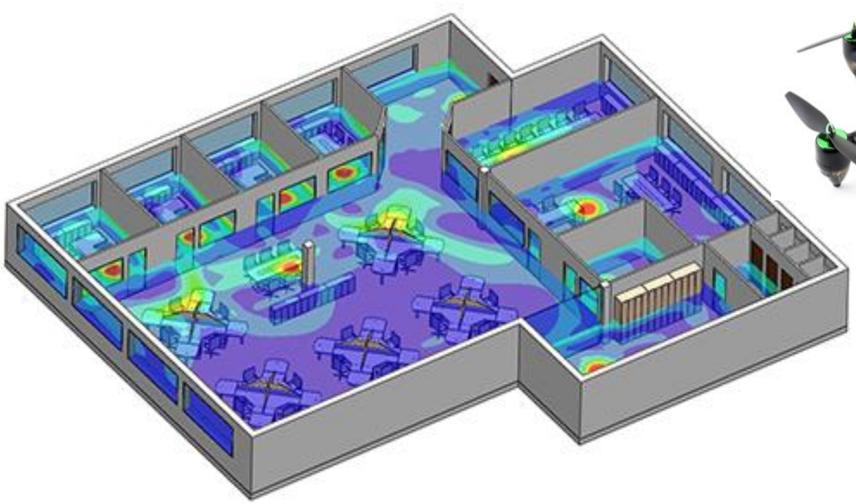


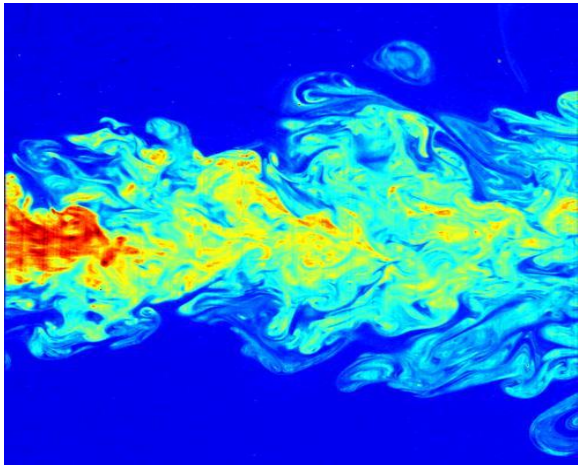
Hengye Yang

Ph.D. Student, LISC

Dec. 21, 2018

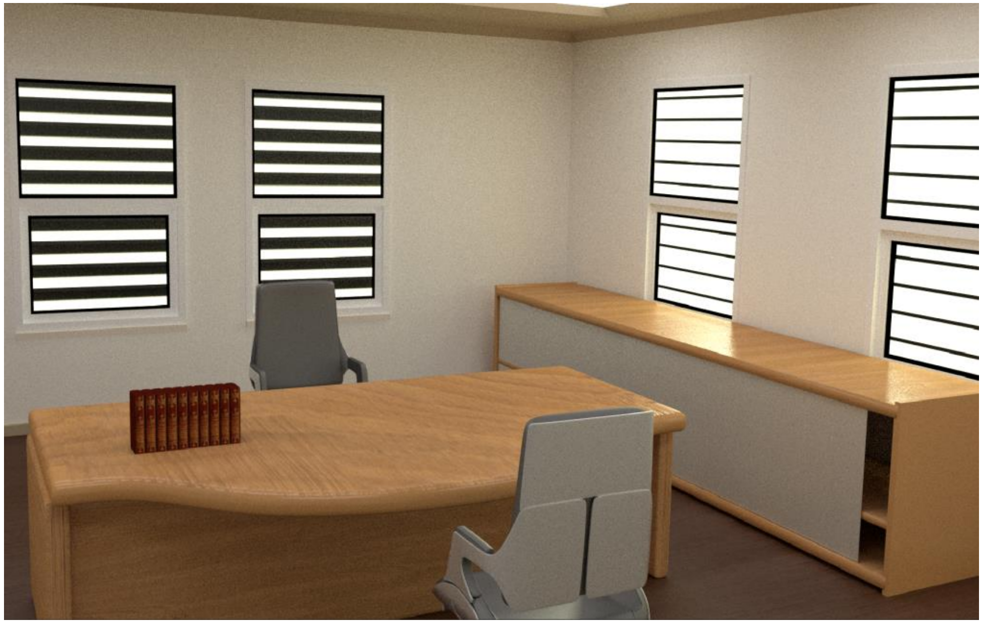
Motivation





- Guide the design of built environment
- Learn about the contaminant transport
- Identify the pollutant source
- Unmanned Aerial Vehicles (UAVs)
- Trajectory tracking and surveillance
- Large wind disturbances

Indoor Airflow Modeling



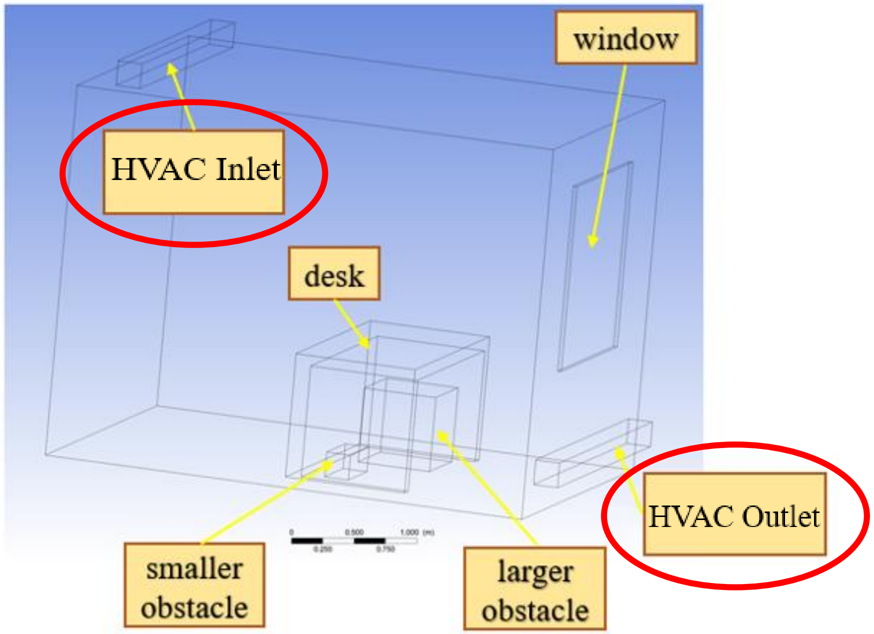
Indoor Environment

- A large proportion of **indoor air pollutants** originates from sources located inside the building, which may cause terrible respiratory and cardiovascular diseases.
- The simulation and forecasting of indoor airflow are of great interests due to its close relationship to occupant's **safety**, **thermal comfort**, and **energy efficiency**.
- Airflow Computational Fluid Dynamics (CFD) analysis is an effective technique for **guiding the design** of new Heating Ventilation Air Conditioning (HVAC) systems.

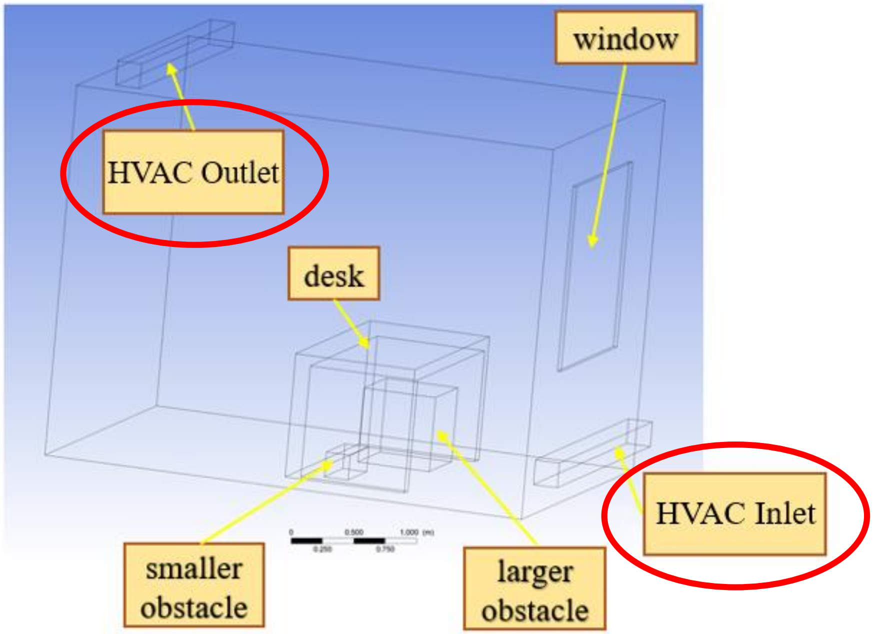
Design Problem

Where should the HVAC inlet and outlet be located?

Which design is more energy-efficient and comfortable?



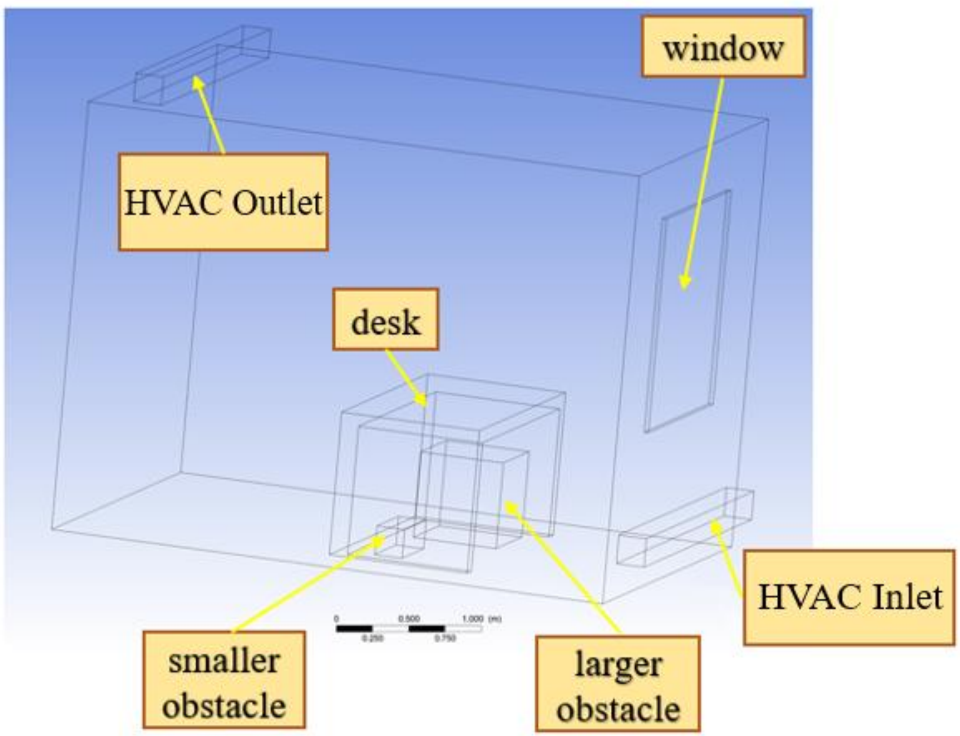
A



B

Indoor Environment

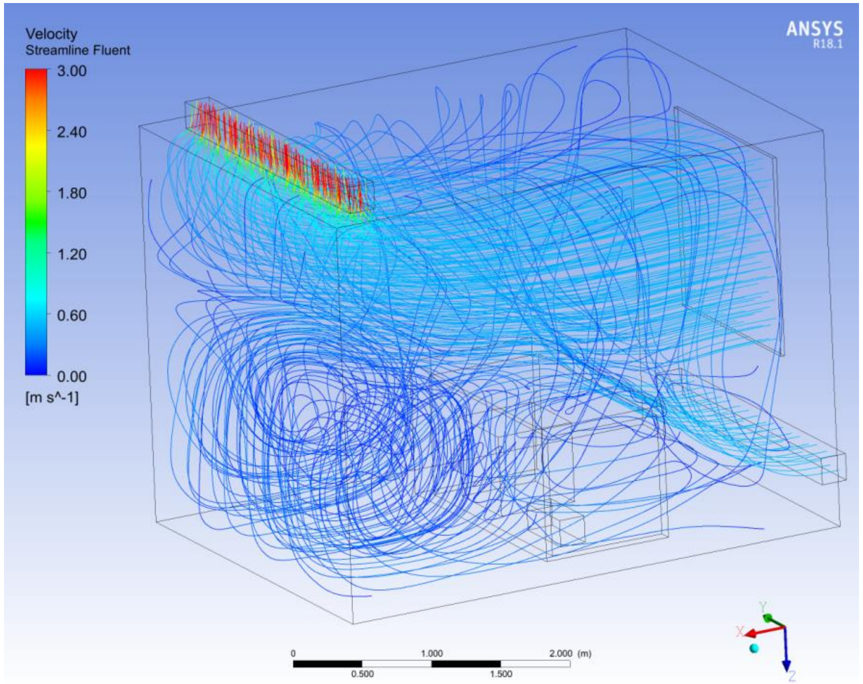
The simulated environment is a 3m × 3m × 4m indoor office space, with a desk in the middle of the room, a window, an HVAC inlet and outlet, and two obstacles placed under the desk.



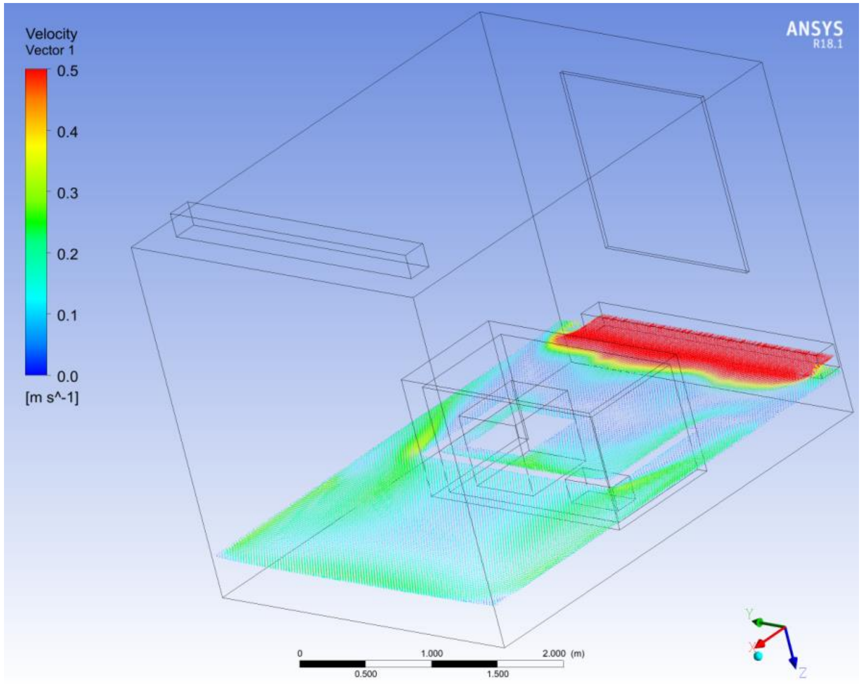
Location	Boundary Condition
HVAC Inlet	Velocity inlet at 0.5m/s
HVAC Outlet	Atmospheric Pressure
Window	Velocity inlet at 0.5m/s

Airflow Simulation

The airflow in the office has been simulated using Computational Fluid Dynamics (CFD). **Streamlines** showing the direction and magnitude of the wind velocity throughout the room and the wind **velocity profile** at the level of the inlet are shown below.



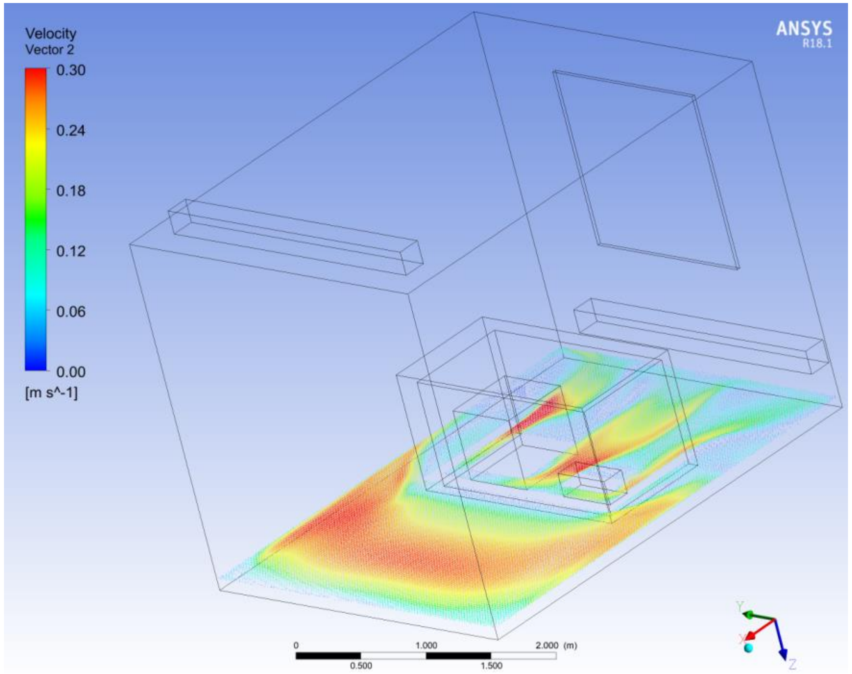
Velocity streamlines



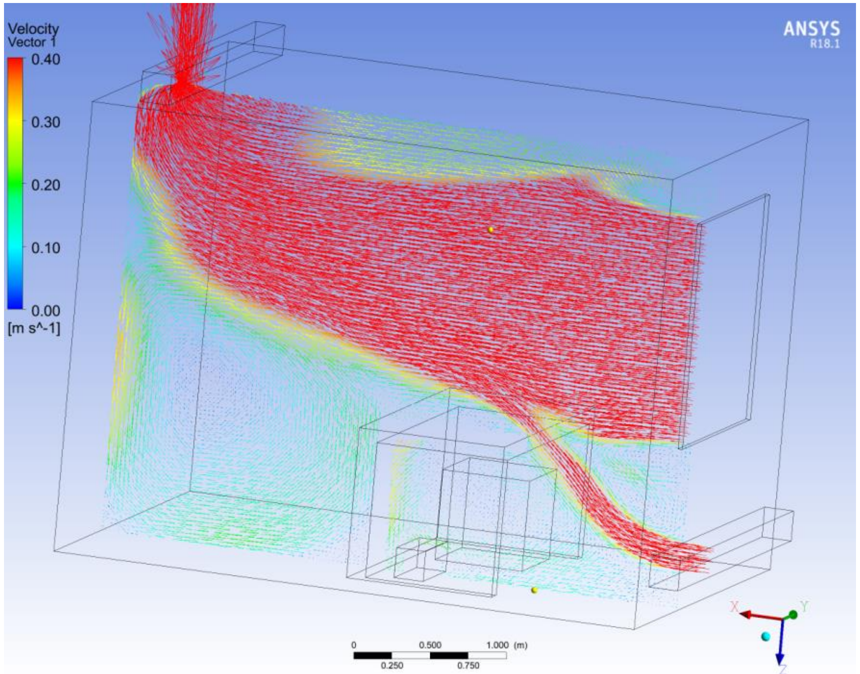
Velocity profile at the level of the inlet

Airflow Simulation

- The maximum wind speed under the table is about **0.3 m/s** between two obstacles.
- Airflows entering the room through the window and the HVAC inlet **mix** with each other above the table.
- Inverse flow occurs between the table and the latter wall.



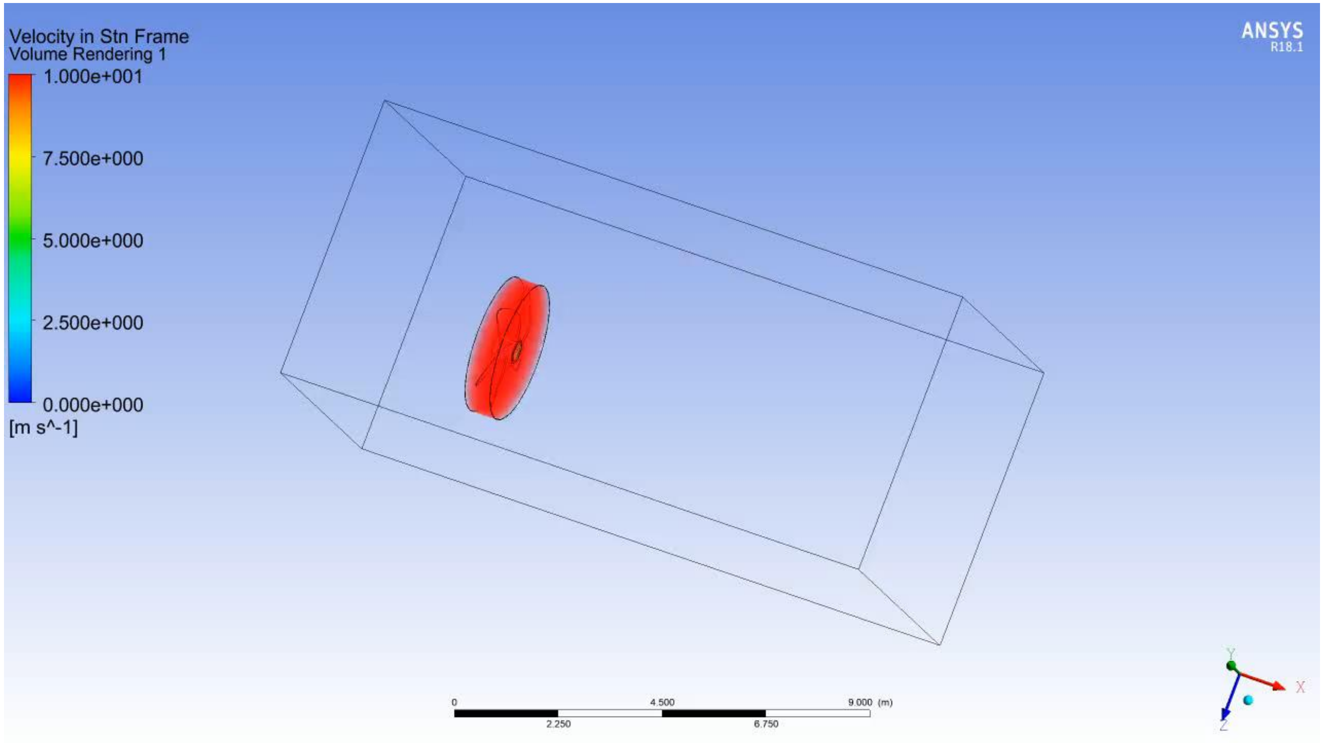
Velocity profile near the floor



Slice of velocity vector profile on X-Z plane

Transient Axial Fan Simulation

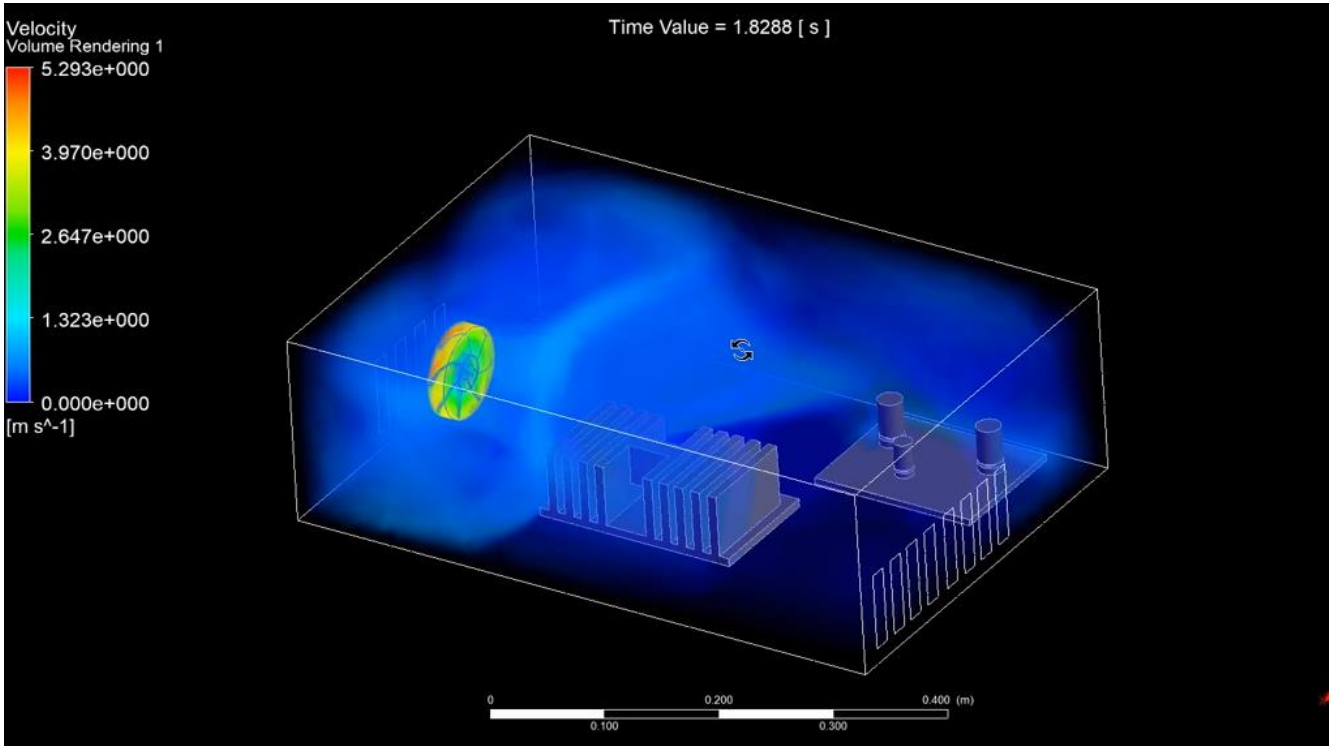
Using the **sliding mesh method**, a one-second transient CFD simulation of an axial fan has been run in ANSYS Fluent. The air inside the enclosure is completely driven by the axial fan and the disturbance is then spread out gradually.



Rendered movie showing the result of the simulation (The color represents the magnitude of the airflow velocity in stationary frame.)

Transient Axial Fan Simulation

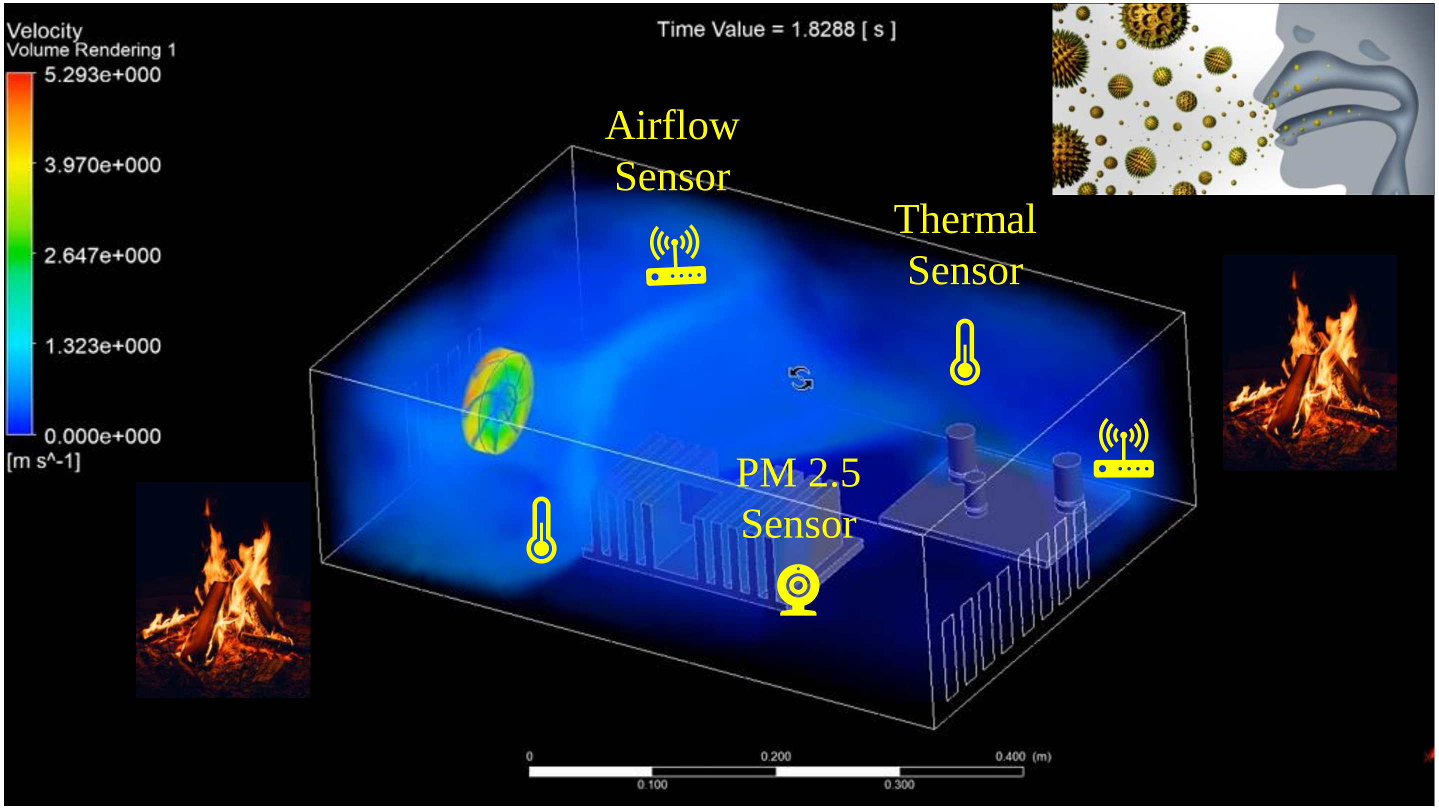
The transient axial fan simulation can then be incorporated into the 3D geometry of an office to simulate an indoor environment with a **rotating axial fan** in it.



If the air distribution does not satisfy the **thermal comfort** and **indoor air quality criteria**, we can use **inverse design methods** to change the thermo-fluid boundary conditions until the design criteria are met.

Data Assimilation

With more complex buildings, system characterization is more challenging. Data assimilation that augment prior knowledge with **sensor data** that monitors building operations to learn about **airflows, temperature field or contaminant transport** may increase the accuracy of the indoor environment simulation and the overall system performance.



Questions?

Thank you