UNIVERSITA’ DEGLI STUDI DI PADOVA

Dipartimento di Ingegneria Industriale DII

Corso di Laurea Magistrale in Ingegneria Aerospaziale

Numerical simulations of the air flow generated by forced ventilation in a rectangular room

Relatore: *Dott. Federico Dalla Barba*

Co-relatore: *Prof. Francesco Picano*

Laureando: *Marco Trevisan*

Matricola: *1234057*

Anno Accademico 2021/2022

Abstract

In these two years, due to the spreading of SARS-COV2 pandemics, it has become important to understand the dynamics of ventilation inside closed environments to determine the infection risk. It is known that Covid-19 is a viral infection and the probability to get infected by the virus increases inside closed spaces because saliva droplets, which contain the virus copies that cause the infection, could remain suspended in the air for long times. In this context, it is important to study the quality of the air inside indoor spaces and quantify the efficiency of ventilation systems in cleaning the air inside the environment. The study of the recirculation of the air becomes crucial to evaluate how long it takes to change a certain volume of air. During ventilation, some vortex could form inside a room, caused by the geometry, the ventilation type and temperature. This can affect the air change rate due to the formation of some recirculations. The problem of air ventilation inside a closed environment is numerically addressed in this thesis by solving the 3D Reynolds Average Navier-Stokes (RANS) equations on a rectangular domain. The air conditioning system is represented by setting two rectangular inflows that inject clean air inside the environment. The air exits from an additional couple of rectangular outflows at the opposite side of the room. Two cases are considered: heating and cooling. It is found that a perfect mixing model can adequately represent the air changing rate inside the room, especially in the case of cooling.

Contents

Chapter 1

Introduction

The Covid-19 pandemic has brought out the problematic of air recirculation inside an indoor space which until now it had been treated only for specific places, like hospitals or laboratories, but it is very lacking in others structures as, for example, schools or malls. Furthermore, the problematic of how a pollutant is dispersed inside an indoor space becomes object of study and common interest. Understanding how to obtain an excellent and rapid air exchange is fundamental for guaranteeing healthy conditions in closed spaces.

This thesis considers a square plan room of 43 m^3 in volume, as a common office, with two inlet and two outlet vents placed on opposite wall that produce an air change condition inside the room. Two cases are considered: a heating system and a cooling system. This combination of vents induces a vortex behaviour of the air that modifies the air change time in the two cases, and it will be interesting to understand how, why, and especially if there is a theoretical model for foresee this behaviour, through which the two systems will be compaired.

The problem of air ventilation inside this closed environment is numerically addressed by solving the 3D Reynolds Average Navier-Stokes (RANS) equations on a rectangular domain. The studied room has a well defined and symmetric geometry, that permits the simulation of only half of the room to reduce the computational cost. Since the focus is on the bulk air change rate, and not on the swirling structures, RANS are the best trade-off between the computational cost of the simulation and the reliability of the results. All the simulations have been done using the software Ansys Fluent since it guarantees good calculation efficiency and stability, with a user-friendly interface, and provides reliable results.

The combination of vents is found to induce the formation of a vortex that modifies the air change rate in the two cases. A perfect mixing model is considered to predict the air exchange rate. The results are compared to the two cases addressed numerically. After that, the influence of the inlet velocity conditions is studied. In particular, how this evolves just outside the inlet vent where the effect of viscosity dissipates the air energy slowing down the flow. Another important task is to evaluate how some particles, emitted by a point source in the middle of the room, are dispersed inside the environment, and how much time they stay inside it before outgoing through the outlet vents. This task has been accomplished by the tracking of unsteady, massless particles.

Chapter 2

Diffusion of droplets in a closed spaces

Viral transmission of SARS-COV2 mainly happens through small airborne microdroplets (called aerosol) that transport the viral load emits by an infected person [1]. A single sneeze emits some large droplets that fall immediately and in proximity of the source, but also some small droplets that floats for several time inside the indoor environments and these are the main vector of infection.

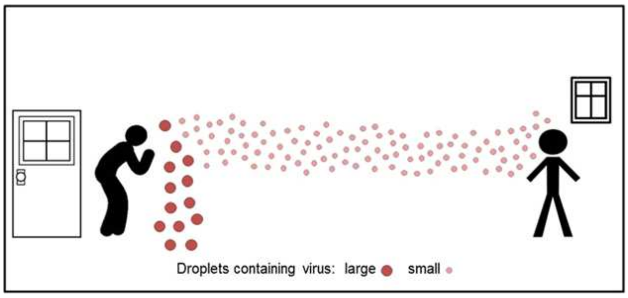

Figure 2.1: Aerosol transmission with large and small droplets [1]

Due to this now there is a new opportunity to contrast the spread of coronavirus. For all the pandemic time we have wore the mask in an indoor environment for reduce the quantity of droplets, but the very important thing is the ventilation for recycle the air inside a room, this reduce the possibility to have a contact whit the viral droplets and ensures the good quality of the air, also not ever air can be changed with a natural ventilation, like open the window, but it's necessary to provide an artificial method composed by an inlet solution with an outlet efflux for air. In this case the situation that will have is a certain type of air vortex in the indoor space but the topic is to know how the air inside the room is mixed with the air emitted by another type of source.

Ventilation is process in which outdoor air is carried inside a space by natural or mechanical means and it is filtered to remove the pollution and the bacteria [2]. The central principle is to remove the contaminated air for substitute it whit clean but the difficult thing is to understand how air is distributed inside a close space (more or less big) and how much of this is possible to change whit mechanical methods.

2.1 Bioaerosol and droplets

The principal way of infection in close space is caused by droplets generated by a symptomatic person whit his exhalations due to a sneeze or when he speaks. Some of these droplets, after some time, evaporate and are converted in an aerosol particles whit relatively small diameter and after becoming airborne. This aerosol particles can infect people who inhale they.

The Sars-CoV-2 can remain viable in aerosol for 3h in the form of droplets [3] and during this period can be infectious but the airborne times be grater if there is a significant cross-flows. Generally, the trajectory of the flow particles depends by the size of droplets and airflow patterns that govern the paths of movement, all of these is governed by the Stokes' that describes the trajectory of the droplets under the influence of the gravity force and the friction. Sneezes and coughs usually constitute a turbulent flow of gas that generates some droplets of various sizes.

Exist a postulate safe distance of 2 meters that represents the safe zone by large droplets ($> 100\ \mu$m). Some new studies explains that the large droplets can fly until 6 meters in a horizontal distance whit a velocity of 6 m/s at the point of expiration. Such scenario in explain in the figure below [4].

In the are represented three different cases of meters when they fall, speed and time:

a. x=6 m, v=50 m/s, t=0.12 s;

b. x=2 m, v=10 m/s, t=0.2 s;

c. x=1 m, v= 1 m/s, t=1 s.

It's possible to see that the speed of particles determinates dimension of the fan which is in a turbulent status. Such status allows the large droplets to

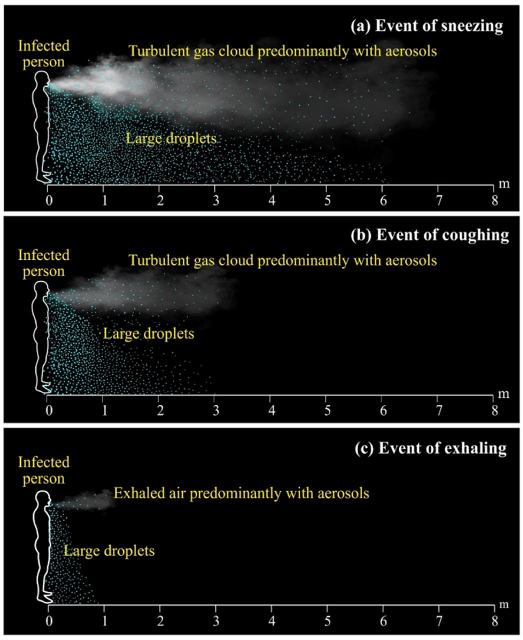

Figure 2.2: Three cases of the trajectory of droplets

increase their distance than the smaller ones.

2.1.1 Simulation of a sneezing event

In a literature is possible to find the results about a simulation of sneezing event, the paper which is cites below is important to understand for having a practical example and some numerical results. The hypothesis for the experiment simplify the situation:

- the jet having the same temperature and humidity as the ambient: T=22°C, RH=50
- there is an inlet condition for simulate the sneezing event.

There are two different evolution's phases after the sneezing event:

i jet phase that is the constant momentum flux when the large part of droplets are emitted;

ii puff phase, this is the later evolution.

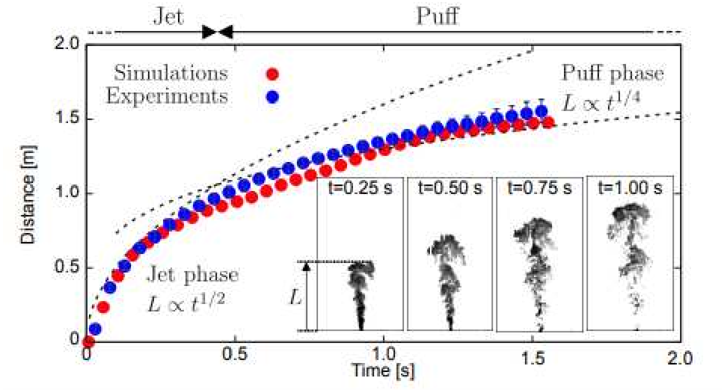

Figure 2.3: A comparison between the experimental and analitical data [5]

It is possible to see that the flux in the first phase, until one second, is compact, but after two seconds by the initial event there is a large dispersion of droplets and the link between distance and time is very different in the two cases, in fact in the jet phase L prop. $t^{1/2}$ while in the puff phase L prop. $t^{1/4}$.

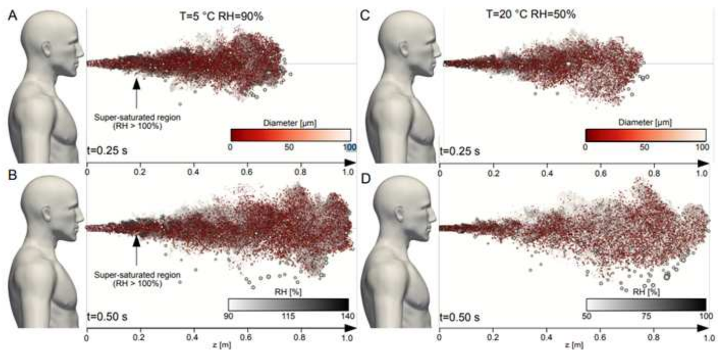

Figure 2.4: Sneezing event [5]

Figure above represents a sneezing event in two different conditions of temperature and humidity. At t=0.25 s, for both cases, the sneeze produces a turbulent cloud saturated with droplets, only a few of these, larger than 100 μm, leaves the cloud and fall. At t=0.50 s there is a dispersion of droplets along vertical directions. The particles dispersion depends on the temperature and humidity. This study confirms that the behaviors of droplets is influenced by the local conditions and even large droplets remain suspended for several time and these remain in the most infection condition. The environment dominates the behavior of flux emitted after a sneeze, in particular temperature and humidity, and these factors also influences the virus exposure, in fact there is a central zone that contains a high level of virus exposure, which is mainly determined by the large droplets.

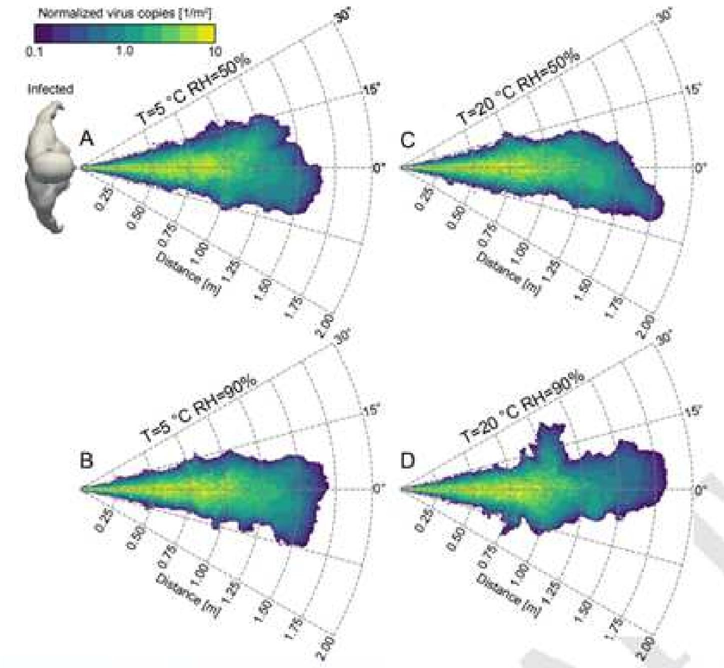

Figure 2.5: Virus exposure inside a close space [5]

Chapter 3

Turbulence

3.1 Turbulence phenomena

Turbulence is a chaotic, statistical and unsteady phenomenon and is governed by the Navier-Stokes equations for uncompressible flow. It can be laminar or turbulent, in the first one it's characterized by some parallel layers and the viscous forces prevail over inertial ones, but, when Reynold's number increase the inertial forces dominate and the motus status changes and become floating whit an increase in resistance.

To define the motus status it has to introduce the Reynold's number that links the viscus and inertial forces because it has a component that represent the viscosity and one of velocity:

$$Re = \frac{UL}{\nu} \tag{3.1}$$

Where U and L are the characteristic velocity and length, while ν is the kinematic viscosity. Normally it can speech about turbulent motus in the case when Re>>5000, in this case there is a strong dependence with the boundary conditions. For this aspect is possible to have an instantaneous field (figure below), but it can change if also the boundary conditions change and is very difficult to determinate it previously, for example for an experiment.

Then is necessary to have more instantaneous fields and for each ones make an average (called *ensemble*), so it's possible to get a statistical field in which it can be predicted the behaviour in the time domain. Now the Navier-Stokes equation could be solved, but they're non-linear so it's neces-

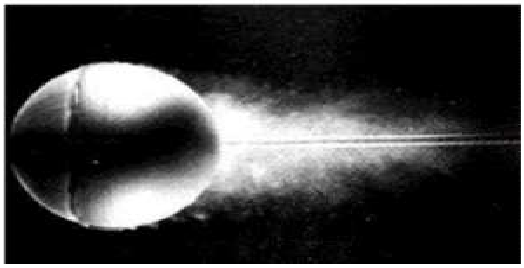

Figure 3.1: turbulence around a body.

sary to find a model of closure. With the introduction of mediated fields, we have a model more like the reality experience, because they show:

- Determinism;
- Symmetry;
- The spatial variations are more similar as the object;
- They are likely to te laminar flux.

There are many eddies, vorticose structures, on different scales, where, the bigger ones, are linked to the characteristic phenomena and the smaller are very intense and localized in where there is the energy dissipation. Each eddy evolves in a vortex governed by stretching and tilting, first consist in the change of orientation and second is change of velocity gradient that leads to becoming the structures narrower in which is large the viscosity term.

$$\epsilon = 2\nu ee \tag{3.2}$$

If we increase the power inside the fluid field, also the vortex structures are increased but their dimensions decreased in order to decrease the dissipated energy.
Energy is transferred to the smallest scales that can't tolerate this quantity of energy and therefore they are closed. This phenomenon is called *Richardson's cascade*. Is possible to calculate the smallest scales with the

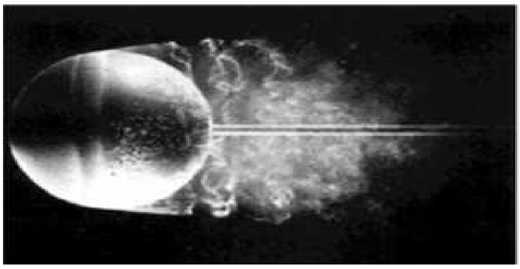

Figure 3.2: eddies around a body.

Kolmogorov theory or K41:

- **Kolmogorov lenght:**

$$\mu = (\nu^3/epsilon)^{\frac{1}{2}} \tag{3.3}$$

- **Kolmogor time:**

$$\tau = (\nu/epsilon)^{\frac{1}{4}} \tag{3.4}$$

- **Kolmogorov velocity:**

$$u = (\nu\epsilon)^{\frac{1}{4}} \tag{3.5}$$

K41's theory provides for three fundamental hypotheses:

- **In the smaller scales** it loses the sense of direction, so the fluid becomes isotropic, homogeneous and universal;
- **Similarity:** if l<< l_0 can depend only on ϵ and ν;
- **If** τ**<<l<<** l_0 solution depends only on l and ν.

3.1.1 Brief notes on fluid dynamics

For solving a fluid dynamics problem occur some important equations and concepts that describe the behaviour fluid field into a region. For this analysis it's necessary to evaluate the motion of an inert particle that is introduced into the fluid field.

The *trajectory* of a parcel is the line that this traces in the space for a time range.

A *streamline*, instead, is the line for which the velocity vector is tangent.

This means inserting a particle inside the room, is possible to evaluate its behaviour in terms of position, velocity and resident time. Then it was considered a point in the middle of the room and it was treat as an injection source of some particles.

A physical phenomenon can be described through in terms of velocity, acceleration, mass, temperature and momentum. If B is one of these dimensions, and b is the value for unit of mass, considering a control volume of fluid, *the Reynolds transport theorem* defines the relationship between the time variation of the volume and the time variation of B. Therefore, it is possible to obtain how much B passes through the control volume and a surface attached to it [6].

$$\frac{DB}{Dt} = \frac{d}{dt}\int_V \rho b\, dV + \int_S \rho b\vec{V}\cdot\vec{n}\, dS \tag{3.6}$$

If the Reynolds transport theorem is applied for fixed control volume and $B = m$, mass, it is possible to obtain *the mass conservation equation*.

$$\frac{D}{Dt}\int_{Sis} \rho\, dV = \frac{d}{dt}\int_V \rho\, dV + \int_S \rho\vec{V}\cdot\vec{n}\, dS \tag{3.7}$$

Considering the time variation of momentum and the sum of the external forces, the second law of mechanics is found that represents the balance of momentum.

$$\frac{d}{dt}\int_V \vec{V}\rho\, dV + \int_S \vec{V}\rho\vec{V}\cdot\vec{n}\, dS = \sum \vec{F}_e \tag{3.8}$$

Therefore, considering the equation of conservation of mass and the balance of momentum, it's possible to obtain *Navier-Stokes equations* which regulate the motion of fluids:

$$\begin{cases} \nabla \cdot \vec{V} = 0 \\ \rho \left[\dfrac{\partial V}{\partial t} + (V \cdot \nabla) V \right] = -\nabla p + \rho \vec{g} + \mu \nabla^2 \vec{V} \end{cases} \tag{3.9}$$

Where:

- $\nabla \cdot \vec{V} = 0$ is the hypothesis of incompressible fluid $\rightarrow$ the density is constant during the motion of the fluid;
- $\rho \left[\dfrac{\partial V}{\partial t} + (V \cdot \nabla) V \right]$ is the material derivative of velocity;
- $\nabla \vec{p}$ is the variation pressure variation along the three axis;
- $\mu \nabla^2 \vec{V}$ is the viscous component of the fluid.

3.2 Large Eddy Simulation

To resolve the turbulence's problem, it may be some numerical approaches because isn't impossible to resolve through direct ways. Two of these techniques are DNS that resolve in every time instance the equation of motus, the second one is the RANS that directly resolve the mean equations. However, both doesn't describe non-stationary phenomena.

So is useful to introduce a new method to resolve the turbulence with the inclusion of the K41 hypothesis, this is a new type of modelling: *Large Eddy Simulation*, LES. The fundamental idea of this method is to study the large turbulence scales, because only these depend by the geometry of field while the smaller ones are necessary only for study their effect to the biggest.

The basic steps to apply this model are:

1. To apply a filter G_Δ to separate the relative velocity for the bigger scale from the lower one;;
2. To resolve the Navier-Stokes equations;
3. To model the unknown terms;
4. To resolve the equations.

G_Δ is the filter operator and delta is the characteristic size of the filter, that is the length which is cut off by the filter. Velocity term it can define with a filtered component $\widetilde{u}$ and a residual one u', the sum of these two terms is the total velocity of the turbulent field. Applied the filter on the Navier-Stokes equations it's obtained the residual stress tensor τ:

$$\tau_{ij}^r = \widetilde{u_i u_j} - \widetilde{u_i}\widetilde{u_j} \tag{3.10}$$

For the K41 theory there is an energy flux that is transmitted until the smaller scale, which in this case is the Δ one, but, for the proprieties of LES method, it's impossible to calculate the energy at this scale, therefore all the energy must be dissipated by a viscosity term, ν.

$$\epsilon = C_s^2 \langle |\delta u|_\Delta^2 \rangle / \Delta \tag{3.11}$$
$$\epsilon = \nu^r \langle |\delta u|_\Delta^2 \rangle / \Delta^2 \tag{3.12}$$
$$\nu = C_s^2 \langle |\delta u|_\Delta^2 \rangle \Delta \tag{3.13}$$

3.3 Reynolds Averaged Navier-Stokes

To simulate a turbulent flow requires a lot of time and computational resources and for simplifying the problem is useful to consider the medium field instead of the instantaneous one so as not to consider the little structures that influence the bigger, increasing the computational difficult of calculus. It's possible to consider an average velocity field to make the motion dependent only on the gradients of the mean field.

Considering the Navier-Stokes equations is possible to apply the Reynolds average:

$$\langle u_i \rangle = U_i \tag{3.14}$$

So the istantaneus velocity field is equal to the average one. Then it gets the RANS:

$$\begin{cases} \nabla \cdot \vec{U} = 0 \\ \frac{\partial \vec{U}}{\partial t} + \nabla(\vec{U}\vec{U}) = -\frac{\nabla p}{\rho} + \nabla \cdot (2\nu \vec{E}) - \nabla \langle \vec{u}\vec{u} \rangle \end{cases} \tag{3.15}$$

In the second equation, the last term, is a symmetric tensor indicating the Reynolds efforts, this needs to be studied in order to solve the RANS. Studying the trace of the stress tensor we realize that it is exactly double the kinetic energy which evaluates the level of turbulent fluctuations in the fluid field.

$$tr(\langle \vec{u}\vec{u} \rangle) = 2k \tag{3.16}$$

After that it is necessary to define how much energy is dissipated in the unit of time, then the term of energy dissipation is introduced.

$$\epsilon = 2\nu \overline{e'e'} \tag{3.17}$$

Practically with the LES method it is possible to see the small swirling structures while the RANS method gives the average fields, like the image below:

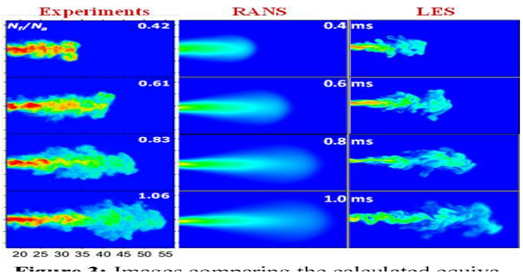

Figure 3.3: Comparison between RANS and LES method [7]

Chapter 4

The model

The goal is to simulate the infection risk inside an indoor place with unnatural air circulation and to see how the infected particles interact with the air flux and the airborne infection risk. It is considered a 4x4x2.7 meters room with an inlet air flow and an outlet one.

To simulate the presence of the pathogen it is introduced a infected people without a mask during his normal breath or while he's speaking, it is expected that the droplets will diffuse in the air following the current lines of the forced air flow, it will then be interesting to understand which areas of the room will be most dense with particles.

The studied model consists in a simple room office with 4x4x2.7 meters dimensions with two inlet and two outlet vents [1].

4.0.1 Core-Care Laboratory

At the Industrial Department of University of Padua was build a climate chamber to simulate the office room with an inlet and outlet conditions. It was divided into four zones.

[1] [8]

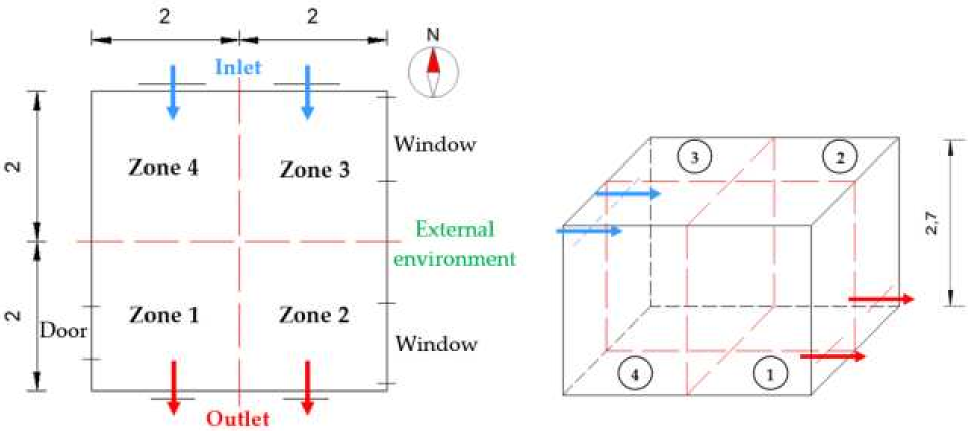

Figure 4.1: Dimensions of the room, [8]

The model is conceived for studying the air change inside an indoor enviroment for three different values of inlet velocity, 1 m/s, 1.5m/s and 2.5 m/s. Inside the room there is a mixture of air called air dirty, while, the inlet vents put inside only clear air, and the model provides a first case whith heating system and a second one of cooling system.

The data about, the air dirty mass fraction inside the room, will be processed for obtaining some graphs which show the emptying and the air change inside the room.

After that, some studies will be carry out about:

- The air's behavior inside the toom, in particular will be studied the vortex;
- The experimental model will be compaired with a theoretical one;
- The behaviour of the air out of the inlet vents will be studied;
- Some massless particles will be released from a point in the middle of the room, for simulating the behaviour of a pollutant.

Characteristics of the vents:

- Dimensions= 10x30 cm;
- Distance from the wall= 65 cm.

Chapter 5

Metodology and results

It was considered a square room with 4x4x2.7 metres with two inlet diffusion in the upper area of the wall and two outlet in the opposite wall. The inside environment was considered as full of a dirty air, that is like a saturated mixture of pollutant and the target is to evaluate how much time occur to change the air inside with a constant inlet of clear air. To simulate the behaviour of the fluid was used Ansys Fluent.

5.1 Turbulent model and mesh

RANS equations consist of the preferred solution model for this case because is not interesting to study the singles eddies, but it is preferable to have a global view of the system's behaviour and also to use, for example, the LES model the computational cost would increase. An advantage in using the RANS equations is the possibility to solve the problem considering only half room without affecting the quality of the results, this because the RANS solves the average fluid field, simplifying the complexity of the calculation.

Studying only half the room allows to simplify the mesh, it's possible to work with a smaller number of elements without compromising the correctness of the results. The mesh is structured in such a way as to have more points in the inlet and outlet zones, for having best results. Structurally it is composed by triangular elements divided by:

- nodes number: 5761;
- elements number: 27111.

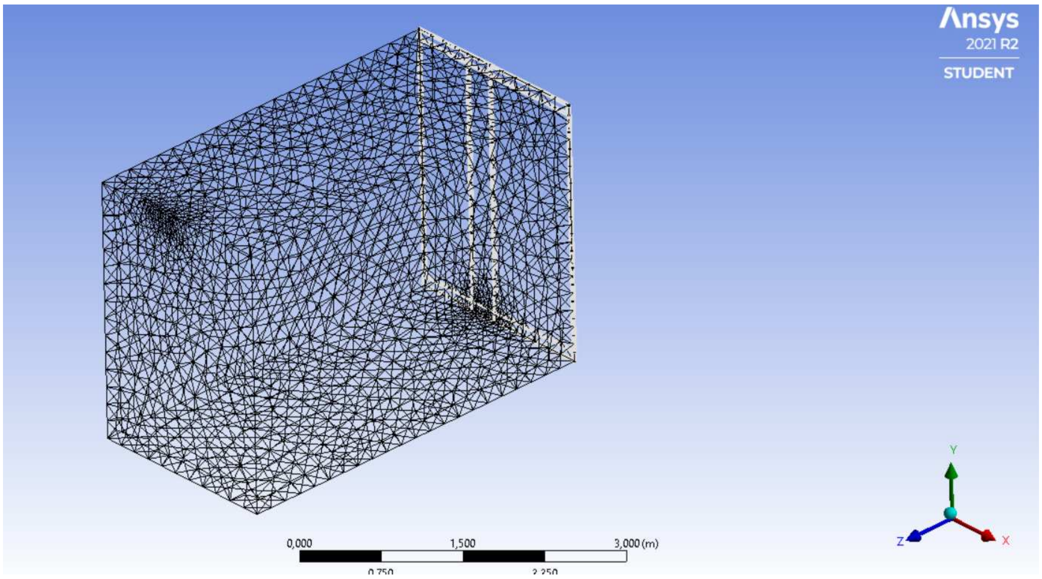

Figure 5.1: The mesh of the model. For the zone of inlet and outlet the points are more for having better results

Ansys Fluent permits to solve the Navier-Stokes equations with the RANS k-epsilon algorithm also considering the transport's theorem. It was considered three different inlet velocity, 1 m/s, 1.5 m/s and 2.5 m/s, that are the most used speeds for a ventilation system. For each of these was studied a heating and a cooling system.

The inlet vents put in the room 100% clean air, while inside the environment there is a mixture of clean and dirty air. For evaluating the quantity of air dirty inside the room during the time was made an average volume integral of the air dirty mass fraction.

5.2 Analytical results

It is possible to have an initial estimate of the amount of air that will be changed in relation to time, knowing only the inlet speed and the volume of the room, applying the mass flow conservation rule:

$$\begin{aligned} \dot{m}_{out} &= \dot{m}_{out,d} + \dot{m}_{out,c} \\ \frac{\dot{m}_{out,d}}{\dot{m}_{out}} &= 1 - \frac{\dot{m}_{out,c}}{\dot{m}_{out}} \end{aligned}$$

$$
\begin{aligned}
&= 1 - y_c \\
&= y_s
\end{aligned}
\tag{5.1}
$$

Now it's possible to link the air dirty mass flow with the total one:

$$
\dot{m}_{out,d} = y_s \cdot \dot{m}_{out} \tag{5.2}
$$

$$
\begin{aligned}
\frac{dy_s}{dt} &= -\frac{\dot{m}_{out}}{m_{out}} \cdot t \\
\frac{dy_s}{y_s} &= -\frac{\dot{m}_{out}}{m_{out}} \cdot t
\end{aligned}
\tag{5.3}
$$

$$
\begin{aligned}
\int \frac{dy_s}{y_s} dy_s &= -\frac{\dot{m}_{out}}{m_{out}} \cdot dt \\
\ln y_s \mid {}^{y_s}{}_{y} &= -\frac{\dot{m}_{out}}{m_{out}} \cdot t
\end{aligned}
\tag{5.4}
$$

By solving the differential equation we obtain the law that regulates the emptying of the room known the inlet conditions:

$$
y_s(t) = y_{s,0} \cdot e^{-\frac{\Phi}{V_0} \cdot t} \tag{5.5}
$$

Where:

- $\Phi = 2vA_{vent} \rightarrow$ volumetric flow rate;
- V_0 is the volume of the room;
- $\dot{m}$ is the area flow rate;
- $y_{s,0}$ is the initial condition, so $y_{s,0} = 1$.

This formula is theoretical and doesn't provide for the role of the temperature, so it is possible that there will be some differences from the results of simulation. After they will be treat in relationship at the two cases, cooling and heating.

5.2.1 Cooling case system

The cooling case system provides:

- an inlet temperature of 290K;
- a room temperature of 300K.

The following images show the final situation of clean and dirty air for the three speeds considered. It is noted that a quantity of dirty air of 20% has been imposed as a sufficient condition.

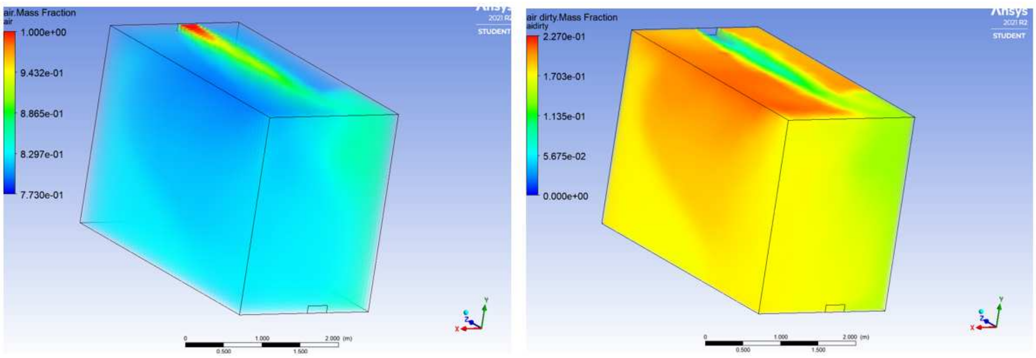

Figure 5.2: v=1m/s

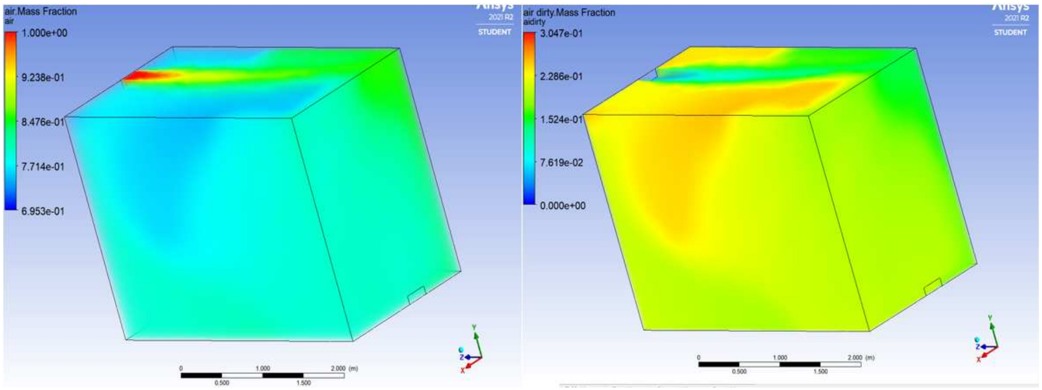

Figure 5.3: v=1.5m/s

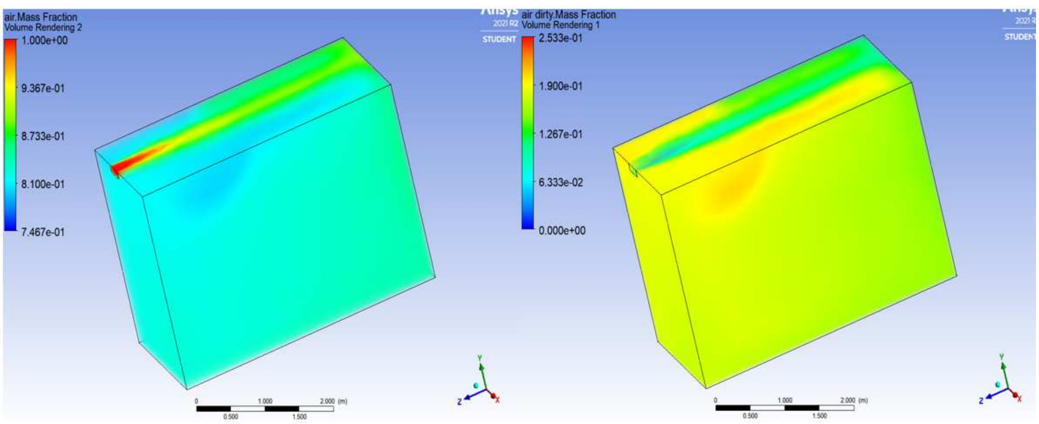

Figure 5.4: v=2.5m/s

For calculating the air dirty volume mass fraction inside the room during the time, it does the volume integral of it and then the results are processed with Excell for obtaining some graph that represent the behavior of the system. At the end of simulation inside the room there is a prevalence of clean air, while the dirtiest one remains a greater concentration in the central area of the room which is a stagnation area, as it is below the two vents.

In the inlet vent zone, there is the maximum concentration of clean air, but, at the end, the dirty one tends to remain in the upper zone, also near the upper edge of the wall. This because, inside the room, the air is warmer than the incoming one and therefore, a part of it, it is push up and has difficulty to get out.

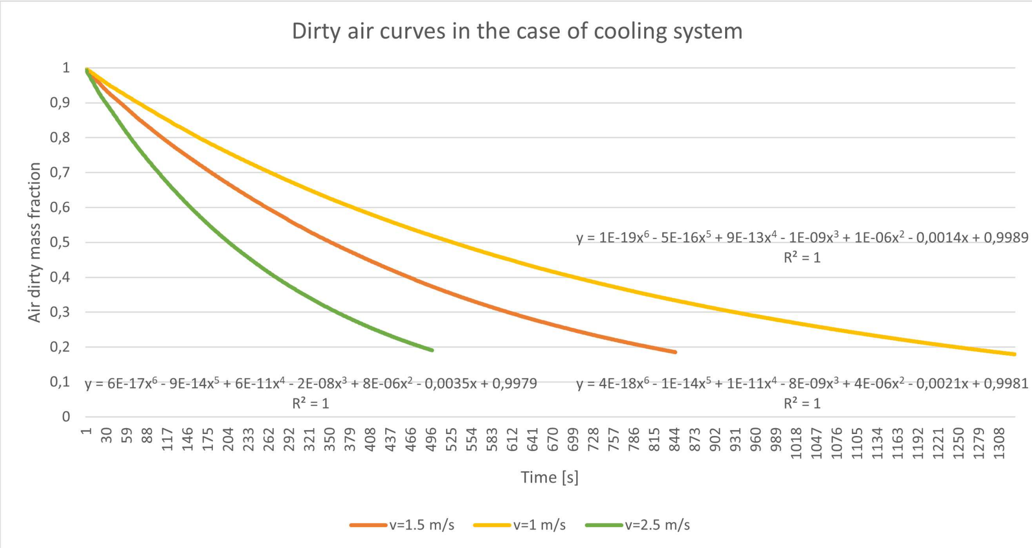

Figure 5.5: Air dirty curves in the case of cooling system

The three curves have a similar behaviour, they are strongly influenced by the inlet speed, and as expected, increasing the speed empties the chamber faster.

5.2.2 Heating case system

The heating case system provides:

- an inlet temperature of 300K;
- a room temperature of 290K.

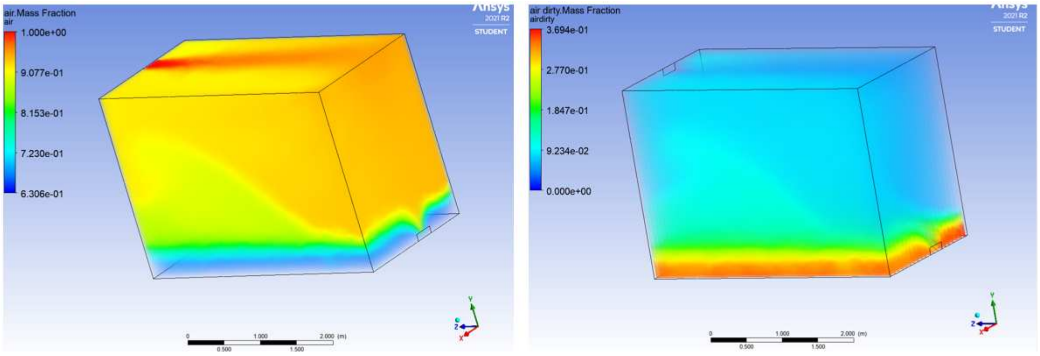

Figure 5.6: v=1m/s

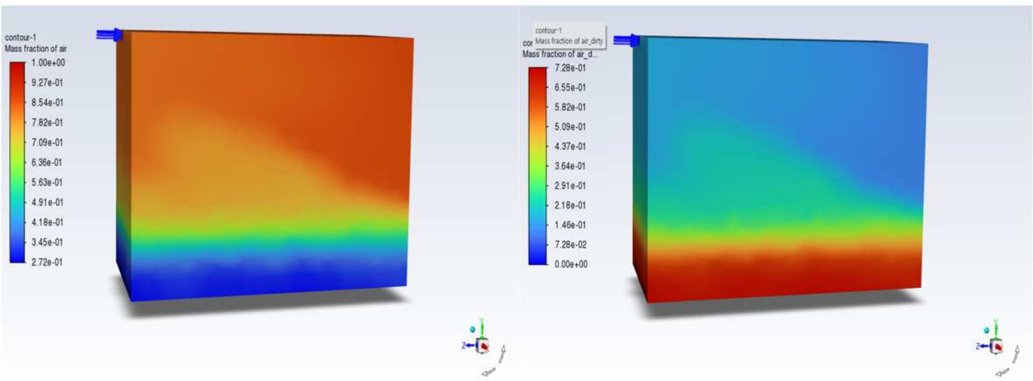

Figure 5.7: v=1.5m/s

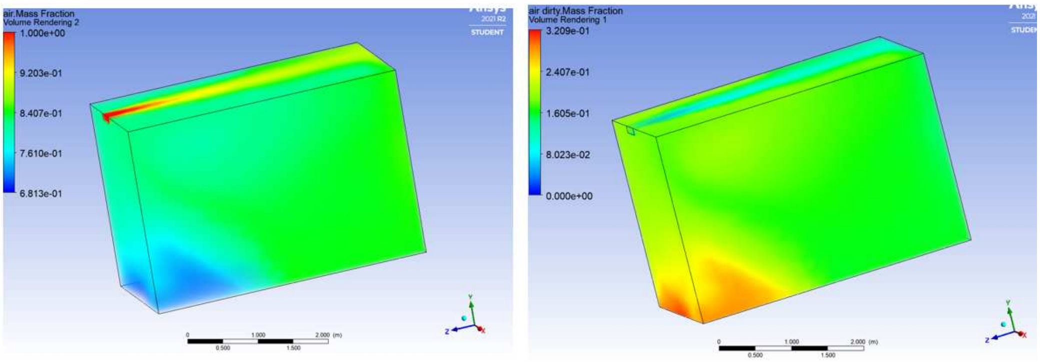

Figure 5.8: v=2.5m/s

The pictures represent a comparison between the clear air and the dirty air mass fraction at the end of simulation. In heating system, the hot ait is immitted with a certain velocity and the difference of temperature between the inlet air and the inside air, produces a vortex condition. The dirty air hardly leaves the room because the swirling structures will push down on the floor zone.

The air which exit from the inlet vents is 100% clear, in fact near the inlet zone it can see how the air behaves, which with the length mixes with the dirty one.

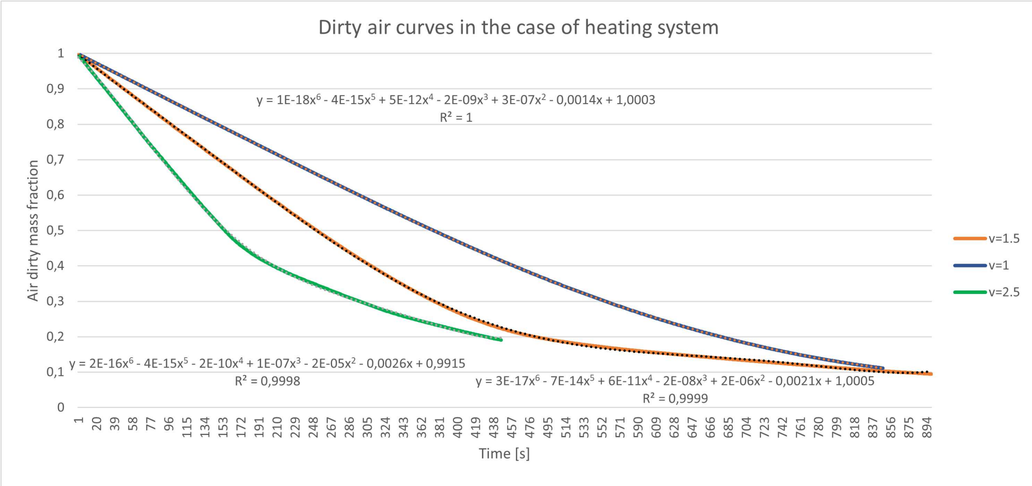

Figure 5.9: Air dirty curves in the case of heating system

From the curves, it's possible to note that increasing the initial velocity, a slope variation is formed for a certain Δt, this fact is more evident in the last simulation. Analysing the images for v=2.5 m/s at Δt=(160,190) s, the formation of a vortex is observed caused by the air recirculation near the wall with the inlet vents.

Trying to do a particle tracker for seeing how the inlet air flow interacts with the inside air. It's possible to note that air, before to exits from the outlet, it makes one or more recirculation inside the room, forming a system of vortices that do not make constant the escape of air from the vents placed at the bottom.

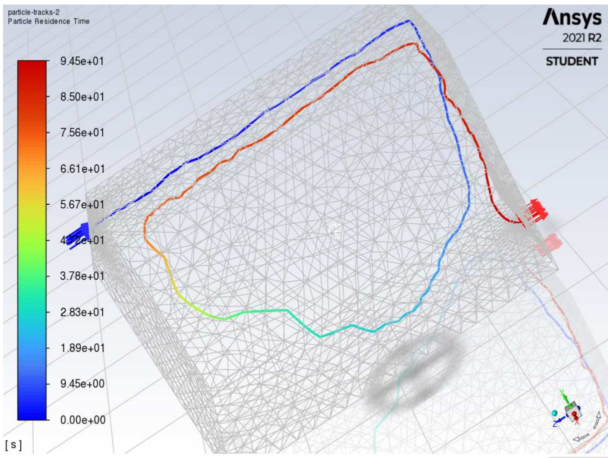

Figure 5.10: Particle tracker from inlet, stream 1

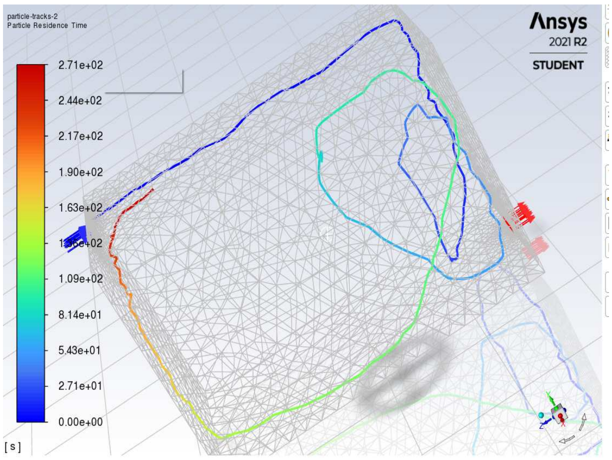

Figure 5.11: Particle tracker from inlet, stream 2

In these two images two different particle trackers are represented and it is possible to see that the air emitted by the inlet vent does not exit directly, but makes some swirls and recirculates many times before going out.

The behaviour of the inlet flux is strictly linked with the boundary conditions, in particular like inlet velocity and room size. For example, considering a bigger room, the behaviour is not necessary the same, but the formation of some vortex is less probably, because the inlet flux it would not immediately hit the opposite wall and it will arrive to it with little kinetic energy.

5.2.3 Comparison between simulation results and theoretical ones

Before the formula for obtaining an estimate of the results was discussed, now that will be applied for having a comparison between the simulations.

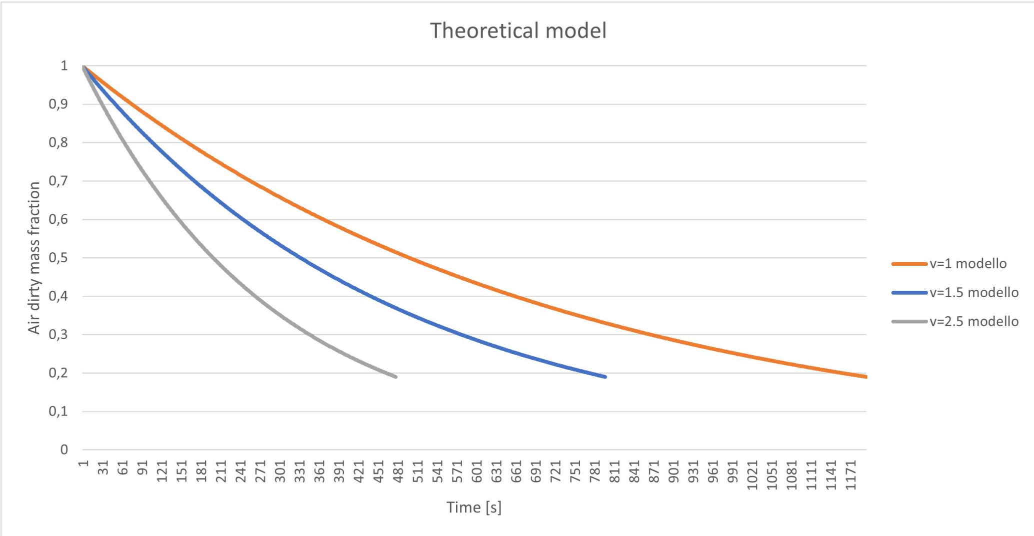

Figure 5.12: Theoretical model

The curve's trend is more similar to the cooling case system because the density of the cool air is bigger than the warm one, and this makes that it goes down quickly stimulating the mixing in the environment. Because of this, from the outlet vents, will exit immediately both dirty and clean air.

Instead, the warm air, is less dense and it stratifies while remain up in the room. However, the air that is immitted pushes out the dirty air that is in the lower zone.

Because of this the cooling system is more similar as the theoretical model, but it's slower as the heating system. Considering like acceptable threshold of ending simulation the 19% of remaining air dirty, the following results are obtained:

Cooling system		Heating system		Theoretical model	
v [m/s]	t [s]	v [m/s]	t [s]	v [m/s]	t [s]
1	1283	1	691	1	1192
1.5	832	1.5	500	1.5	795
2.5	498	2.5	447	2.5	477

5.2.4 The inlet vortex in the heating system

The air flux generated by the inlet vents, produces a turbulent stream that depends by the inlet speed and this one evolves in length and width. The stream flux conditions the behaviour of the air inside the room, in fact, with the heating system case, the difference of velocity and temperature will generate an air vortex below the inlet vents, in the middle of the room. This whirling system causes the stagnation of air, at a certain time period depending by the initial speed, that increases the air change time.

For studying this phenomenon it's focused on the inlet area, dividing the vent through a horizontal plane placed in the middle of it and parallel to the ceiling. It was considered four vertical lines, parallel to the vent, posed to a difference distance from the inlet, and also a fifth horizontal line perpendicular from them. Now it's important to study the variation of velocity and temperature in this region for having a scheme for what concern that region.

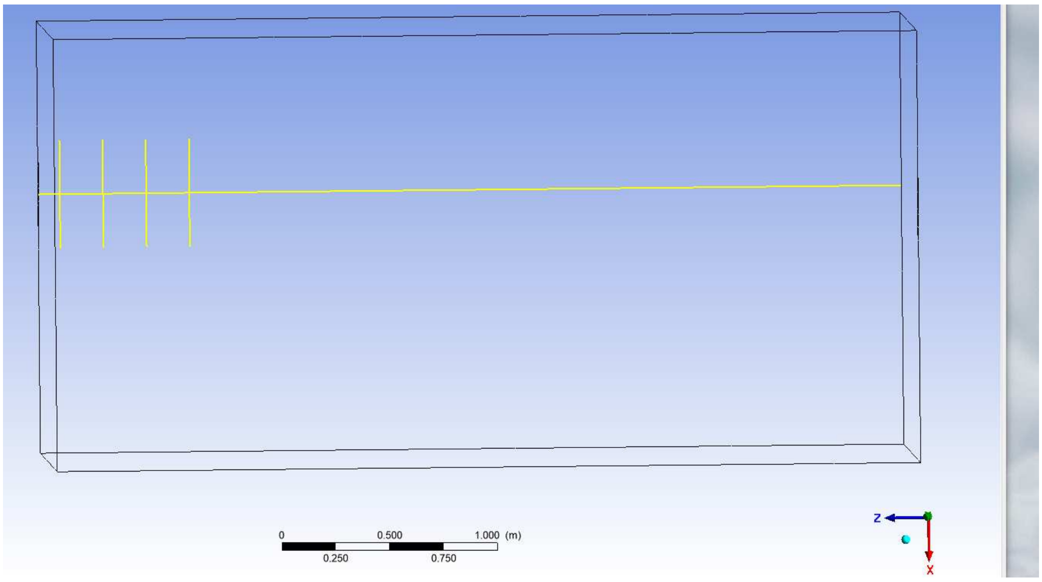

Figure 5.13: The scheme for the study about inlet

The target is to study how the velocity and the temperature evolved just outside the vent. Both can be expected to disperse increasing the distance, due to the interaction with the air present inside the room, which is certainly still.

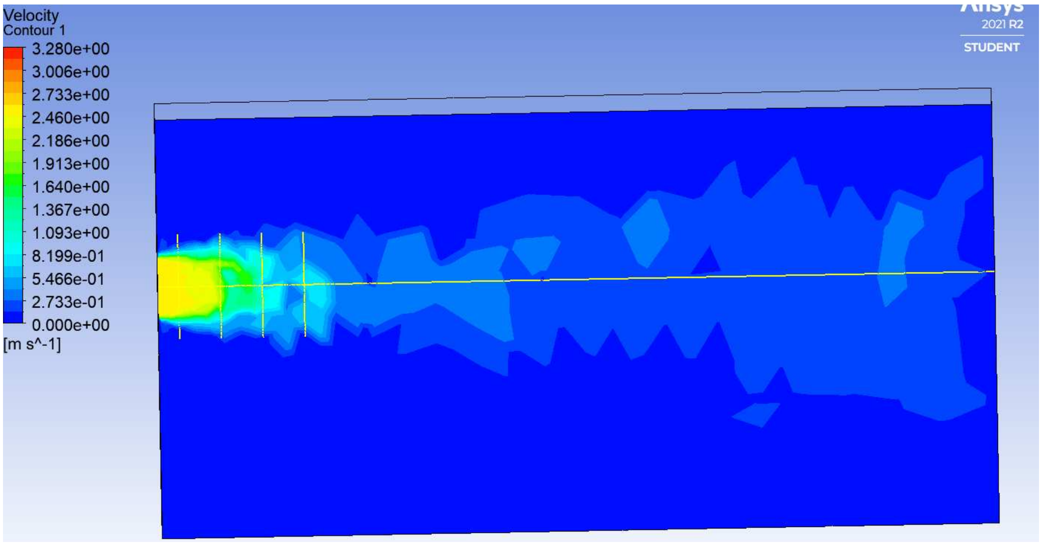

Figure 5.14: Velocity distribution out of the vent

The entry speed, in this study, is 2.5 m/s, it immediately tends to decrease due to the effect of viscosity and small dissipative vortices that form in the outermost regions.

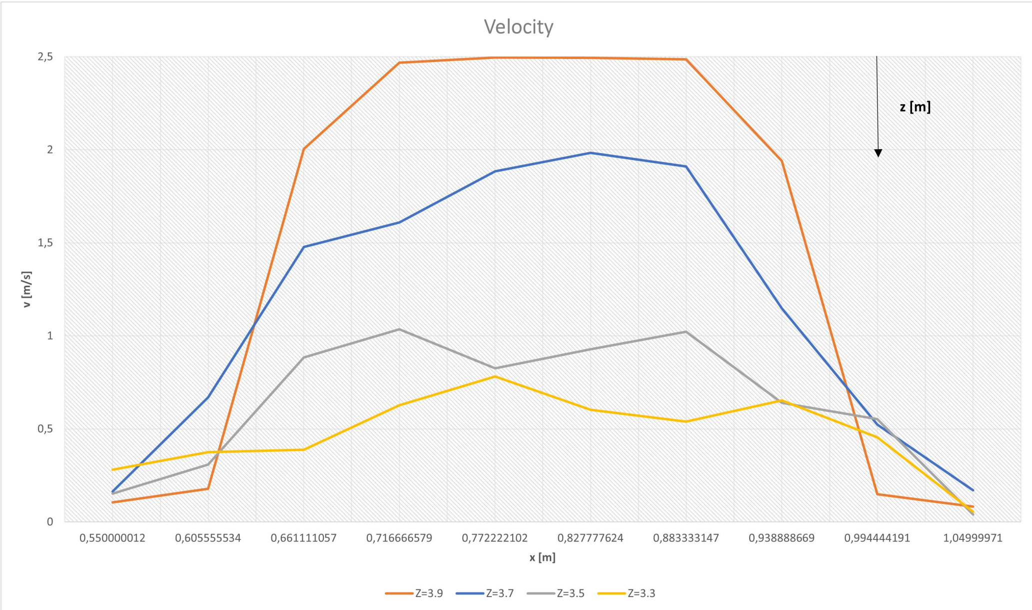

Figure 5.15: Velocity distribution along the vertical lines

The orange line represents how is the flux just a little out of the vent. The fluid field is more compact and is not affected by the viscosity effects, in fact the curve is initially constant in its central part, and there is a sharp change in speed at its extremity and then rapidly decays to zero.

At three centimetres from the inlet the speed becomes unsteady, and decreases compared to before, the first fluctuations begin to arise inside the fluid field, *blue line*. In fact, the velocity isn't constant growing and decreasing in a totally random way enough to provoke the formation of an air recirculation.

The grey and the yellow line are respectively five and seven centimeters far from the vent, at this distance the initial rate of velocity is a half, the effect of viscosity has dissipated the kinetic of the flux that arrives nearly static to the opposite wall (image below). Now the flux is not compact as before, but it is open like a fan.

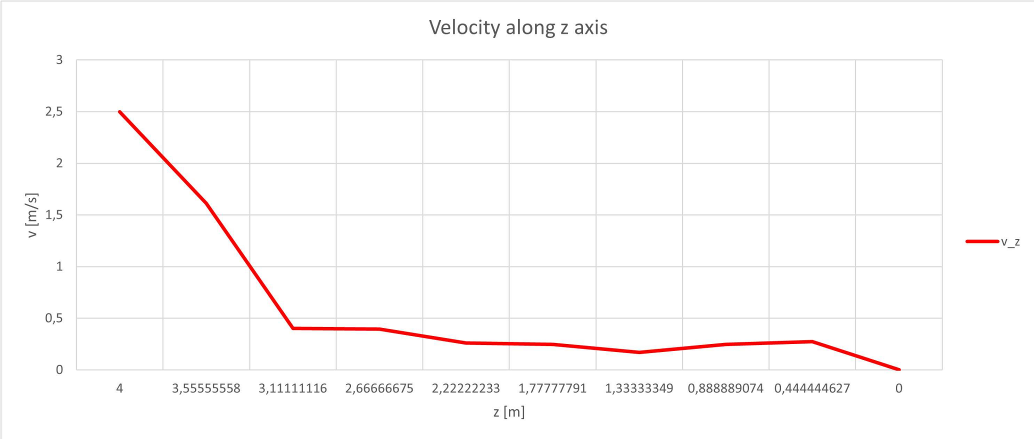

Figure 5.16: Velocity along z axis

For what concern the temperature's distribution it can be made the same considerations as the it has a same behaviour as the velocity except for the central zone [1] of the flux. Here the temperature is quiet constant, except for some small variation, at the end it will mix with the air inside the room and it will reach the condition of equilibrium.

[1]For central zone means that between 0.7-0.9 cm

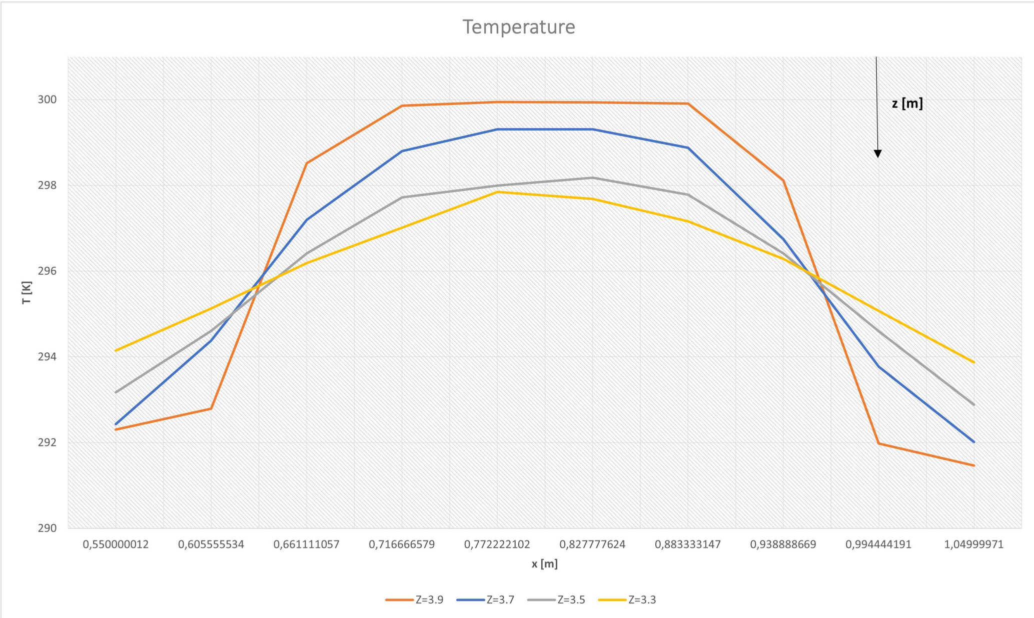

Figure 5.17: Temperature distribution along the vertical lines

5.2.5 A particles dispersion from a generic point

Inside an indoor space it's important to know how some particles are dispersed in the environment in case of dispersion of a pollutant.

Considering a point placed in the middle of the room from which some particles were injected inside the room. For studying a totally general case, the particles were considered massless, that is they are only a tracer that moves according to the flow conditions inside the room.

These particles were emitted only for the first ten seconds of simulation, after that, the dispersion is evaluated in the subsequent time's instants, until any of these will be out.

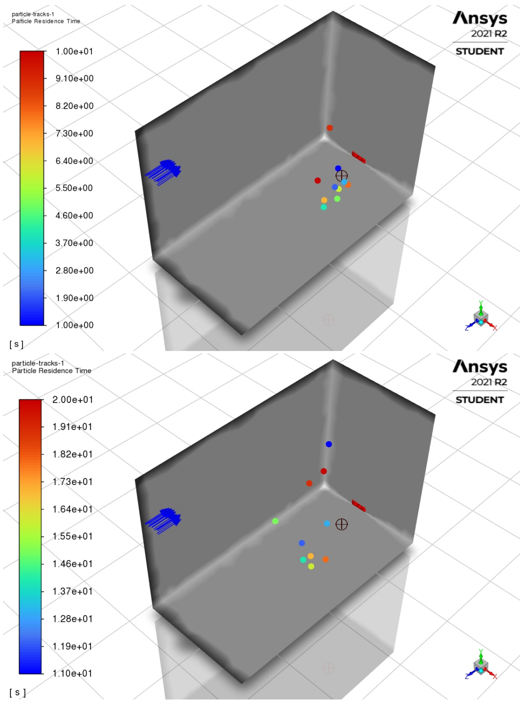

Figure 5.18: Particles distribution after 10s and 20s

Now there are ten particles inside the room, at t=10s, the last one is emitted. They are all near the emission point, like a cloud, except for the first emitted that is already moved away from the point. After another ten second, at t=20s, the particles begin to disperse in a random way based on the air flux.

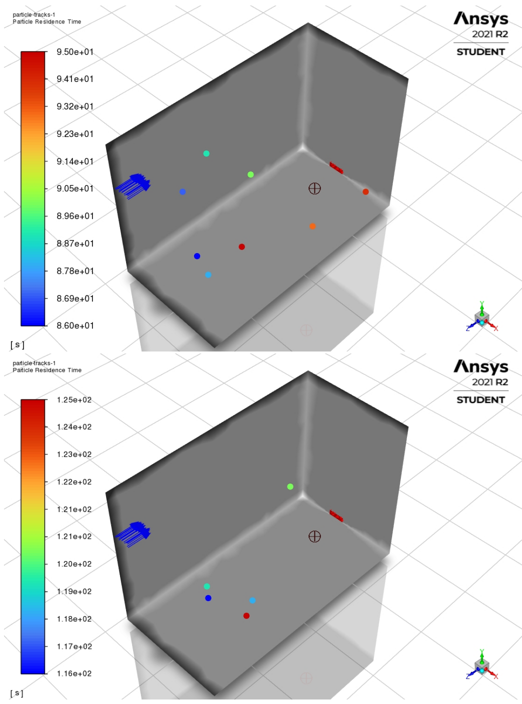

Figure 5.19: Particles distribution after at t=95s and t=125s

At t=95s the particles begin to exit by the outlet vent, at this moment they are eight and after another thirty seconds they are only five. This means that the particles stay inside the room at least for ninety seconds.

To use a massless type of particles could be reductive for describing the problem of dispersion of pollutants, but it is a useful method for having a first interpretation of the problem.

If a simulation with inert particles is used, that is, with a certain mass and diameter, it's possible to have the gravity effects which push down it.

Chapter 6

Conclusion

This thesis considers the study of the ventilation system of a square plan room of 43 m^3 in volume, as a common office, with two inlet and two outlet vents placed on the opposite walls of the room that guarantee the air change inside the environment. Two different cases are considered: a heating system and a cooling system.

The problem is numerically addressed by solving the 3D Reynolds Average Navier-Stokes (RANS) equations on a rectangular domain using Ansys Fluent. RANS are the best solution for this problem in terms of computation time, stability of results andsimplicity of the mesh. A perfect mixing model is considered to predict the air exchange rate. In particular, the injected air is considered to instantaneously mix with the air inside the room. The predicted emptying curves are compared to the two cases addressed numerically obtaining a good agreement.

The cooling system presents more similarities to the theoretical model than the heating one. Indeed, the density of the cold air is bigger than the warm one and this makes the air go down quickly enhancing the mixing in the environment. Because of this, from the outlet vents, will exit immediately both dirty and clean air. On the other hand, the warm air is less dense and tends to stratify while remaining up in the room. The air that is immitted pushes out the dirty air that is in the lower zone. This causes additional difference from perfect mixing conditions and is the reason why the perfect mixing model performs better in the cold case.

The air flux near the inlet vent is strongly influenced by viscosity. In fact, the speed decreases away from the vent, generating some small swirling structures which slow down the fluid.The last task addressed in this thesis is the study of how some massless particles interact with the room environment. Inserting a point source in the middle of the room, ten particles were emitted from it for the first ten seconds of the simulation. Initially, they stay near the injection point, but, subsequently, they tend to leave from it dispersing in the air volume. The particles stay inside the room about for ninety seconds before the first exits from the environment

Bibliography

[1] Lidia Morawska, Junji Cao. Airborne transmission of SARS-CoV-2: The world should face the reality. *Enviroment International*, 2020.

[2] Covaci Adrian. How can airborne transmission of COVID-19 idoors be minimised? *Enviroment International*, 2020.

[3] Mahesh Jayaweera, Hasini Perera, Buddhika Gunawardana, Jagath Manatungea. Transmission of COVID-19 virus by droplets and aerosols. *PMC, National Library of Medicine*, 2020.

[4] X Xie, Y Li, A T Y Chwang, P L Ho, W H Seto. How far droplets can move in indoor environments. *PubMed*, 2007.

[5] J. Wang, M. Alipour, G. Soligo, A. Roncon, M. De Paoli, F. Picano, A. Soldati. Short-range exposure to airborne virus transmission and current guidelines. *Thesis*, 2021.

[6] G. Graziani. Aerodinamica. *Università La Spazienza*, 2018.

[7] S. Simbedu, et all. Simulating Flame Lift-Off Characteristics of Diesel and Biodiesel Fuels Using Detailed Chemical-Kinetic Mechanisms and LES Turbulence Model. *Proceedings of the ASME 2011 Internal Combustion Engine Division Fall Technical Conference*, 2011.

[8] M. Marigo, G. Tognon, M. De Carli, A. Zarrella. A zonal model for assessing the infection risk distribution of COVID-19 in indoor environments. *Department of Industrial Engineering, Università degli Studi di Padova, Italy*, 2021.