Multiscale approach for the study of ventilation system in Frèjus tunnel in case of fire

Politecnico di Torino, Department of Mechanical and Aerospace Engineering
Corso Duca degli Abruzzi, 24 – 10129 Torino, Italy

Supervisor:
Vittorio Verda

Candidate:
Fabrizio Peiretti

A.Y. 2019-2020

Index

List of figures

List of Tables

Abstract

The complexity in the analysis of fire evolution requires many years before the complete definition of a device able to simulate and predict its behaviour. Since the evolution of numerical simulations, there was the necessity to create a software able to analyse critical conditions generated by fire studying how control it. Many aspects must be considered such as the reaction of the species or the effects of the external environment on the fire itself because they can change its evolution in time. The case study considered in this research is the Frejus tunnel. It is one of the longest transalpine tunnels with a transversal ventilation system due by its length. The history shows many critical accidents connected with fire inside of closed environments as for instance in underground stations or, as in this case, in tunnels. One example is connected with the accident happened in the Mont Blanc tunnel where many people lost them life. Moreover, one of the things that is highlighted studying older accidents is that people in danger are not only who is directly connected with the vehicles enrolled in the accident itself, but many times one of the worst cause of death is connected with the intoxication caused by the smoke produced by the fire and propagated along the tunnel. For these reasons the smoke evolution after fires becomes a very important point in the research because it is possible at least to limit and control the effects generated by an accident trying to confine as much as possible the smoke and as a consequence also the temperature in the tunnel. The main goal of this job is to focus in a deeper way on the analysis of the fire evolution in Frejus tunnel caused by a burning vehicle inside of it trying to define certain guidelines about how the ventilation system has to react in order to confine the propagation of the smoke.

The simulations are performed with the help of a particular method called *Multiscale approach*. The basic software that is used is Fire Dynamic Simulator (or easily FDS) which is a software born specifically for the analysis of fire in closed environments. It allows to simulate a 3D space with a fire inside and through analytical equations implemented in it, it is able to describe the evolution of fire and of the smoke with the possibility to add control systems in order to study also the transient evolution of the system. The software is based on the common Navier-Stokes-Fourier (NSF) set of equations with the addition of other physical laws such as the ones connected with combustion processes. The main problem of FDS is connected with the real time of the simulation that is required to end one process. In

fact, the higher is the number of cells the longer will be the time to obtain the results because the software has to perform a higher number of calculations. This is why multiscale approach is introduced in simulations of this type because it can avoid problems connected with time. This is also the reason why a modified version of the software is adopted instead the original one because the original version of FDS does not allow any connection between the 3D and the 1D model. The basic concept at the base of this procedure is that instead of analysing the entire tunnel as a 3D space with a very high number of cells, the overall length of the Frejus tunnel can be split in three regions: two far zones and one near zone. As the name of the zones suggests, the near zone will be the area in which fire is located during simulations and for this reason it will be studied with a 3D model with FDS software because precise informations about the fire evolution must be obtained from that. The dimensions of the 3D zone depend on some parameters assumed at the beginning of the simulation such as the maximum power of the fire or the dimensions of the tunnel. About the far zones, they are instead just the connections with the external environment in the case of Frejus tunnel and they are considered as simple 1D elements attached to the 3D mesh because no effects of fire are present there. In these areas there could be accelerators or fans but no smoke and no fire can be simulated. This technique makes the software incredibly faster with respect the overall computational time of the entire tunnel with a 3D approach. The resolution of the 1D space is so fast due by the fact that it is based always on the NSF equations but it uses an algorithm called as *simple* that through an iterative procedure decreases incredibly the time to get the resolution of the equations. The far and the near zones communicate through a continuous exchange of boundary conditions passing information related to pressure, temperature and velocity between the surfaces of the 3D space and the nodes of the 1D network attached to them.

After a brief introduction of the evolution of safety in tunnels, it is possible to analyse using the software previously described the Frejus tunnel with respect different initial configurations. The main important parameter analysed during the simulations is the speed of wind inside. Due by the length of the Frejus tunnel, the environmental conditions at the two portals can change affected by different variables such as temperature, density, humidity and so on. For this reason, it is necessary to consider different case studies in which the pressure difference changes: depending on the pressure difference between the portals, the direction of the wind will change and also the wind speed will be affected by that. The ventilation system has to react against to this natural ventilation generated depending on the x coordinate of the vehicle. The desired effect of a reaction of the ventilation system is to make as much as possible a zero velocity section in correspondence of the fire location in such a way that the smoke is completely confined where the fire takes place and there are no areas in which the smoke is still without any extraction possibility or even better avoiding smoke travelling along the tunnel. It is not possible to present a final law that can be used to control the system in order to limit the diffusion of the smoke in every possible situation inside of the tunnel because a large number of variables can create completely the situations (the position where the fire takes place, the power of the fire, the wind velocity, the traffic conditions). The configurations that should be studied to provide a complete analysis on the ventilation system are many and for these reasons it is preferred to focus only on some cases trying to touch all the possibilities that can be found.

What is presented is just a summary of the results obtained after many tests. Some tables where the results of the simulations performed are listed depending on the parameters set at the beginning are presented. For sure the results that will be proposed are not the unique solution admissible for each configuration. A fire located in the middle of the tunnel can be controlled considering different solutions of the ventilation stations upstream and downstream with respect the fire itself.

The overall results about the ventilation strategies are performed considering something similar to a control system but it is not a real retroactive device. The control phase of the ventilation system is not so easy to be performed due by some limitations of the software. In fact, the modified version of FDS does not allow to use the common list of commands to control the variables connected with the 1D model. For this reason, it is possible to control dampers present inside of the 3D space just adopting smoke detectors connected with them. On the other hand a different strategy is adopted to apply something similar to a control system also on the fans far from the fire zone that is not a real retroactive system as in the case of dampers, but it could be the base of some improvements.

The problem treated in this thesis is very complex and there is still a large part of research focused on it due by the complexity of all the configurations that can be done. Many studies are still in progress in order to improve the numerical simulation methods for instance with the addition of these control devices able to automatically tune the ventilation system parameters basing directly on the properties recorded by the 3D environment.

Nomenclature

c_p	specific heat at constant pressure [J/kgK]
D_h	hydraulic diameter [m]
e_{tot}	specific total energy [J/kg]
e_i	mass source or sink [J/kg]
e_k	mass source or sink [J/kg]
e_m	mass source or sink [J/kg]
$\vec{g}$	gravity [m/s^2]
l_m	mixing length factor [m]
m	mass [kg]
p	pressure [Pa]
q	heat transfer [J/kg]
Re	Reynold number [-]
T	averaged temperature [K]
T’	temperature fluctuations [K]
u	longitudinal velocity [m/s]
u’	longitudinal velocity fluctuations [m/s]
U	averaged longitudinal velocity [m/s]
$\vec{v}$	velocity vector [m/s]
V	control volume [m^3]
w_k	mass source or sink [kg/m^{3}s]
α	thermal diffusivity [m^2/s]
Δ	low-pass filter [m]
ε	kinetic energy density [J/kg]
λ	thermal conductivity [W/Km]
μ	dynamic viscosity [Pa s]
ν	cinematic viscosity [m^2/s]
ρ	density [kg/m^3]
τ	stress tensor [Pa]

Objective of the elaborate

The work on which this thesis is focused is connected with the analysis of the ventilation strategies that should be adopted in case of fire in Frejus tunnel. The simulations performed are just focused on some particular cases because the number of factors that can affect this problem are incredibly high such as the thermodynamic properties of the environment, the traffic conditions, the evolution of fire. A first preliminary step is done in order to describe a little bit how the software works in order to understand better also the results. One important factor that must be considered is connected with turbulence. There are still many mathematical methods that can be used in order to deal with it and a brief introduction on them is performed in order to present which kind of process will be adopted to study fire problems.

The real important part of the job is focused, as said before, on the analysis of the ventilation system of Frejus tunnel. The complexity of the structure and also the length of it, requires making a deep analysis on the safety plant in order to avoid terrible accidents as in the past happened. The transversal ventilation strategy is one constrain imposed directly by the length of the structure because if the fire for instance happens in the middle, it is not safe at all to extract the smoke through one of the two portals giving to the smoke the possibility to travel for more than 6 km. If a longitudinal ventilation strategy is adopted for the case study for sure it will be dangerous for both rescue teams and users of the tunnel because many times the cause of death in the tunnel accidents is connected with intoxications and at least them can be limited controlling the smoke evolution with the ventilation system of the tunnel considered. During the simulations, it will prove that the diffusion of the smoke is strictly connected with the speed of the wind inside of the tunnel. This natural wind is caused by different levels of pressure difference between the Italian and the French portals due by the environmental thermodynamic properties at the two extremities. The pressure differences considered during the simulations are mainly focused on the evolution of fire. From previous studies performed by Carvel, it is possible to prove that also the fire evolution is affected by the wind velocity inside of the tunnel. Three different levels of pressure are considered because they are able to give the fastest growth of the fire in the middle of the three main branches of the structure as will be better explained in the following chapters. The Frejus ventilation plant can be subdivided in three main branches equal in dimensions and controlled by six ventilation stations. The way in which it is created is really influenced by the need to control the terrible effects that an accident can cause if happens. All the six extraction or injection points must be tuned in order to confine as much as possible the smoke propagation in the tunnel. Both the extraction and the additional supply of fresh air must be controlled in case of fire in order to confine as much as possible the smoke closed to the area in which fire takes place. For this reason, the main parameter on which the analysis will be focused is the speed of the air inside of the 3D space. What is desired is to tune the ventilation stations until a zero-velocity section is obtained in the region where the simulated burning vehicle is present.

As mentioned before, it is not possible to analyse all the configurations because there is an infinite number of possible cases that could be found in reality. Some of them are presented where some of these variables are supposed at the beginning (traffic conditions, position of the vehicle in key locations and others) but for sure it is not possible to define a complete law of how the system has to work because each case must be analysed with its own peculiarities.

1. Introduction

1.1 Basic fluid-dynamic equations

The computational fluid dynamic method or CFD is a way in which it is possible to solve real physical problems with the help of computers through the implementation of particular algorithm in them. Usually, fluid-dynamic problems are solved with the really known Navier-Stokes-Fourier set of equations. In this case, this system of equations is characterised simply by the three most important properties able to describe the motion of one fluid which are the continuity, the momentum and the energy involved in the motion of particles. So, they are simply balance equations obtained through the analysis of a single cell selected as a small portion of the body that is analysing. Below it is presented in a simple way how the set of equations is obtained just because it is the fundamental point that characterise all fluid-dynamic problems.

The continuum assumption is a basic concept of the analysis of the fluids. This means that even if the fluid from a microscopic point of view is characterised by an infinite number of particles, in this case it is considered as a continuum matter with properties that can be described in a very small portion of volume and varies from volume to volume [1]. As a consequence, each property is a function not only of time but also of space. Through the continuum assumption, it is also possible to define that:

$$\rho = \lim_{\Delta V \to 0} \frac{\Delta m}{\Delta V} \tag{1.1}$$

So, it means that for infinitesimal portion of the volume it can be defined as a constant value the density in it. A simple concrete explanation of what is this continuum assumption can be done just considering that the matter in general is composed by atoms that are very small particles which constitute the objects around us. What the relation (1.1) explains is that these atoms cannot be considered singularly, but it is necessary to evaluate the material considered as a continuous matter even if in reality it is not because many different particles constitute it. This is the reason of the presence of that constant density at very small portion of volumes. This is the first approximation that guides through the definition of the three equations used to define the well-known Navier-Stokes-Fourier set of equations.

The first that is defined here is the continuity equation which is obtained just with a simple balance on a hypothetic small cell which characterises a portion of material. This equation describes the change of the mass inside of the small volume considered and for this reason the mass flow rate is the property that can do this. The mathematical formulation of the continuity equation is characterised by two parts where the first is describing the quantity analysed and how it varies in time while the second is the term that is called as advective flux and it describes the amount of the particles considered that are going out from the control volume bringing with them a part of the quantity analysed that in this case is the mass. Moreover, another relevant aspect is that in this case no connections between the lagrangian and eulerian point of view are done because this is not the aim of this elaborate. An additional justification of that is connected with the fact that many times the Eulerian point of view is easier to be used to analyse the problems because it allows to consider a fixed control volume and after that analyse how it communicates with the external environment. In the case of the lagrangian point of view, it is required to define a control mass and after that study the evolution of them which is not so easy especially in the case of a tunnel where the eulerian method is easier to be applied. For a better explanation on the difference between the two methods it is recommended to have a look at that elaborate [2]. For this reason, all the following equations are presented considering the eulerian point of view which is of interest for the application that will be exposed after. After a brief explanation of what are the two points of view about how to study the system, the first simplest equation referred to the continuity can be presented below.

$$\frac{\partial \rho}{\partial t} + \nabla \cdot \rho \vec{v} = 0 \qquad (1.2)$$

In this case no internal sources are considered because otherwise it is no more possible to talk about the continuity equation, but it is called as balance equation where the formulation is the following one:

$$\frac{\partial \rho}{\partial t} + \nabla \cdot \rho \vec{v} = w_k \tag{1.3}$$

The term on the right-hand side describes the possible sources or sinks that can be present inside of the volume considered. An additional consideration can be done for some particular fluids. In the case of uncompressible fluids, for sure the density will not vary in time and as a consequence, if no other sources or sinks are present in the control volume, it is possible to confirm that the divergence of the speed is equal to zero [3].

$$\nabla \cdot \vec{v} = 0 \tag{1.4}$$

This property will be used many times to simplify the equations which describe the problem because in this way one possible variable is completely removed making it as constant. For instance, during the analysis of the case study, once computed the density of the air that flows inside of the tunnel, for all the calculations it is considered as constant.

About the second important equation instead, the principle connected with conservation of the momentum says that momentum is conserved till external forces interact with the system considered changing it. In general, two types of forces must be considered which are:

- Body forces: applied on the particles inside of the small volume considered and for this reason they are defined per unit of mass (gravity, electromagnetic forces, …);
- Surface forces: applied on the surface of the same volume but this time they are defined with a unit per surface (pressure, viscous forces).

Moreover, it is necessary to also add the net rate of momentum due to convection that in this case is described by the dyadic product in the formulation below.

$$\frac{\partial \rho \vec{v}}{\partial t} = -\nabla \cdot \rho \vec{v}\vec{v} - \nabla p - \nabla \cdot \tau + \rho g \tag{1.5}$$

Where the pressure is a scalar and the τ is a tensor which describes the behaviour of the small volume in the three dimensions with respect stresses. In particular the division between these two tensors has also a physical meaning because the pressure tensor embeds all the terms connected with the normal forces that are acting along the three main directions of the small body and as a consequence perpendicular to the control volume faces. Normally it can be also written simply as the scalar quantity p times the identity matrix because when the volume considered is too small as in this case, it is possible to say that the pressure does not change on the three main directions and it is almost constant even if the position is not the same and for sure the environment pressure will change. Otherwise, the second stress tensor embeds all the shear terms that are present on the small volume without considering the normal stresses. In this case the formulation for Newtonian fluids it becomes as follow:

$$\tau = -\mu(\nabla\vec{v} + \nabla\vec{v}^T) + \left(\frac{2}{3}\mu - \kappa\right)(\nabla \cdot \vec{v})I \tag{1.6}$$

Once again in the case of uncompressible fluids, the second part becomes null and the equation is simplified really much. Newtonian fluids are the simplest ones because they show a constant viscosity independently on the velocity at which this property is measured.

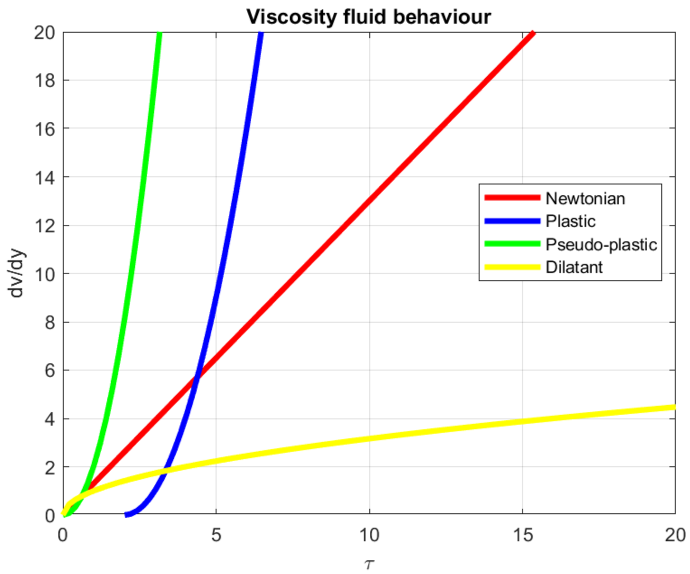

Figure 1.1: Fluid viscosity behaviour depending on shear stress.

As can be seen by the graph, the Newtonian fluids show the easiest behaviour with respect the change in speed from the point of view of the viscosity. There are also other kind of behaviours that in this case are not presented because less relevant and used just for specific applications. For the case studied in this relation, it is convenient just to focus on the Newtonian fluids because they describe the behaviour of some fluids as water, air, alcohol, glycerol, benzene and motor oil which are the fluids of interest for the application that will be studied. For an explanation of the behaviour of non-Newtonian fluids it should be referred to [4].

The last of the three main equations which characterise the fluid-dynamic phenomena is the energy equation. Before to explain the relation which describes the energy balance, it is necessary to make a distinction on all the possible source of energies that can be found. This is really easy because the overall energy of a system is characterised by the summation between the energy present internally of the system itself and the mechanical one and the latter is subdivided by two more components which are the kinetic and the potential part (to have a clear explanation of this, [5] is recommended).

$$e_{tot} = e_i + e_m = e_i + e_k + e_p \tag{1.7}$$

Once known this concept, it is not so difficult to obtain also the equation connected with the rate of change of the total energy because the main idea is to connect once again the flow of total energy present in a fluid-dynamic problem and the interactions with the external environment. Due by the fact that the total energy is defined as a composition of both thermal and mechanical parts, for sure the external forces that must be considered are both. In the end, the final equation of the total energy will be something as follow:

$$\frac{\partial(\rho e_t)}{\partial t} + \nabla \cdot (\rho e_t \vec{v} + q + pI\vec{v} - \tau\vec{v}) = 0 \tag{1.8}$$

The first term in brackets is connected with the advective flux which means the mass of particles that are bringing part of the total energy outside or inside of the volume considered. The second one is instead connected with the external thermal sources or sinks that interact with the small portion of volume called also as control volume. That term is quite relevant because allows to solve the final set of equations and it is known as Fourier law expressed as:

$$q = -\lambda \nabla T = -\rho c_p \alpha \nabla T \tag{1.9}$$

The formulation is exactly the same but it depends on the kind of parameter that is known because λ is called as thermal conductivity and it is measured as $\left[\frac{W}{mK}\right]$while the second α is called thermal diffusivity and it is instead measured as $[\mathrm{m}^2/s]$. Finally, the last two terms are easily defined through the well-known stress tensor previously described in the momentum equation. From this equation it is obtained the internal energy knowing the definition provided about the mechanical energy.

The three equations explained in the previous parts, can be used together to make the complete solution of a fluid-dynamic problem.

$$\begin{cases} \frac{\partial \rho}{\partial t} + \nabla \cdot \rho \vec{v} = 0 \\ \frac{\partial \rho \vec{v}}{\partial t} = -\nabla \cdot \rho \vec{v}\vec{v} - \nabla p - \nabla \cdot \tau + \rho g \\ \frac{\partial (\rho e_t)}{\partial t} + \nabla \cdot (\rho e_t \vec{v} + q + pI\vec{v} - \tau \vec{v}) = 0 \end{cases} \tag{1.10}$$

These three equations allow to close the problem because considering also the two additional formulations provided by Navier with the definition of the shear stress tensor and by Fourier with the law which describes the thermal heat:

$$\begin{cases} \tau = -\mu(\nabla \vec{v} + \nabla \vec{v}^T) + \left(\frac{2}{3}\mu - \kappa\right)(\nabla \cdot \vec{v})I \\ q = -\lambda \nabla T = -\rho c_p \alpha \nabla T \end{cases} \tag{1.11}$$

All of these equations are just at the base of a real fluid-dynamic simulation but they are fundamental. Moreover, they are referred only for Newtonian fluids which means that they are true only for fluids where the viscosity is not a function of the velocity of the fluid itself as explained before. In addition, this is the system focused simply on thermodynamic and mechanic of the problem without considering the chemistry behind it. In the software all the equations related to a more precise model are implemented but the aim of this introduction is just to present the procedure adopted to solve the problem without reaching that level of precision as it is in the software explained in the following chapters. The last thing to notice is that those expressions are presented in the more general way. Sometimes, it could be convenient to decrease the number of unknowns considering that a fluid-dynamic problem can be simplified from a three-dimensional problem to a one-dimensional as will be done considering the multiscale approach.

1.2 Analytical methods to describe turbulence

The need of a software able to simulate the behaviour of the fire and not only, but also the possibility to simulate the behaviour of the smoke and how it propagates during an accident of this type was really important from the point of view of safety. Even if the mathematical model was developed many years ago, it was not so easy to create a software like FDS able to solve in an easy way fluid-dynamic problem of this type. The problems were mainly connected with the fact that the technology was not able to create computers able to make those complex and strong calculations and also because the complexity of fire and smoke evolution is not so easy to be described with mathematical equations. For these reasons, some simplifications and some assumptions must be done when the code of this software was defined as suggested in [6] and [7].

Turbulence is present almost everywhere in reality of things. Many ordinary phenomena present strange effects when they occur such as the mixing phase between two fluids. Turbulence is one of the trickiest parts of the fluid-dynamic analysis because it is presented as a real irregular succession of vortexes and eddies. From the point of view of the mathematical model, it was required more than one century between the Navier-Stokes-Fourier set of equations for a generic fluid-dynamic problem and the first mathematical models able to describe with numerical equations what is happening in the motion of a turbulent flow. Turbulence cannot be neglected when the study of the fluid motion is performed because it affects in a relevant way the fluid evolution in time. Moreover, as in the case of fires for instance, it is necessary to be able to estimate it in order to predict in a better way the behaviour of the fire and of the smoke and how they propagate in order to design a proper ventilation system inside of the tunnel for instance. Turbulence is not so easy to be analysed due by the complexity of the phenomenon and also by the fact that it is a nonlinear dynamic problem.

Turbulence was a really well-known phenomenon but, as said before, it is not so easy to describe it. Reynold was the first scientist interested on the analysis of this phenomenon starting from the famous experiment that bring him to the definition of what today is known as the Reynold number [8].

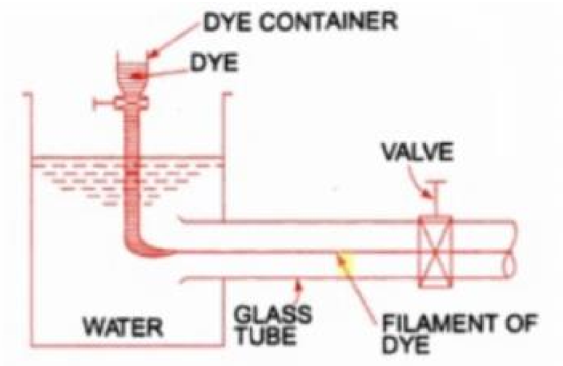

Figure 1.2: Simplified scheme of Reynold experiment.

This famous experiment of Reynold was very easy. In order to analyse the motion of a turbulent flow, a dye flow motion was analysed in such a way that the distinction with the water around of it was very clear. Reynold made this experiment many times and what he noticed is that changing the speed of the dye, the effects that he saw were very different each other. In particular, in the case of very slow dye, it was possible to see the perfect laminar flow without any effect of turbulence but when the speed of the dye increases a little bit above a certain threshold, turbulence starts to appear until the flow is really spread and a huge amount of eddies and vortexes are present when the speed of the fluid is increased very much. For this reason, Reynold created a factor able to describe the behaviour of the fluid saying if it is a laminar or turbulent. The coefficient is defined starting from the momentum equation because in the case of an uncompressible fluid as it is, it can be possible to define the momentum equation considering that the divergence of the speed is null. Moreover, in the case of the Reynold experiment the flow can be considered as unidirectional and for this reason considering that the $\vec{v} = v(u, v, w)$, only the u coordinate is relevant. In the end the momentum equation can be written as:

$$\frac{\partial \rho u}{\partial t} + \nabla \cdot (\rho u \cdot u) + \nabla p - \mu \nabla^2 u = 0 \tag{1.12}$$

The Reynold number is simply defined as a ratio between the advective part and the viscous term. This means that the advective part causes instability in fluid motion and as a consequence turbulence while the viscous effect present in the properties of the fluid make it much more stable reducing the effects of turbulence [8].

$$Re = \frac{\rho u L}{\mu} \tag{1.13}$$

The last thing that was defined after this experiment is the ranges of passage state of laminar flow and the other of turbulent flow:

- Laminar: $Re < 2100$;
- Transition: $2100 < Re < 4300$;
- Turbulent: $Re > 4300$.

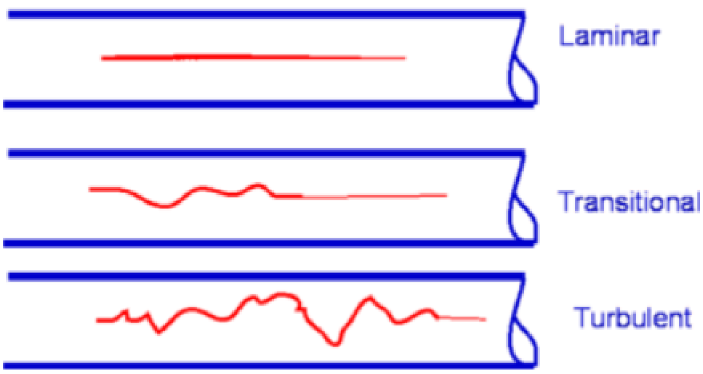

Figure 1.3: Fluid behaviour obtained by Reynold tests.

Even if it is possible to describe a certain range in which turbulence is verified and even if the Navier-Stokes-Fourier set of equation is very well proved in description of the motion of a fluid, it is not so easy to find a connection between the deterministic way in which the NSF equations are written with respect the apparent random motion of the turbulence. Many theories are defined in order to make possible the connection between the strange motion of the turbulence and NSF equations and below some of them are described making a strong analysis on the method adopted by the software used to make the fire simulations.

An interesting observation of the Reynold experiment is that if the test is made many times with the same exact conditions, the properties measured such as speed or temperature do not coincide at each infinitesimal time step. Instead what remains the same are the averaged properties that are almost the same. For this reason, it can be possible to say that for sure a turbulent flow presents a particular sensitivity to very small perturbation in the system such as a small perturbation of the boundary conditions, of the initial conditions or of the material conditions. It is possible to prove that even if a very small change of these three aspects is applied, there will be a completely different result on the final developed flow.

For instance it is not so easy to have exactly the same material for all the parts that constitute the experiment because the production phase of that material provide to it some different imperfections for sure that will change the result of the turbulence. This aspect must be considered very well because the possibility to focus just on the averaged flow instead of having a particular solution of the equation in each point of the grid in the case of turbulent flow is one possible solution.

The problem of turbulence was developed for a very long time before to get the results that are present nowadays. The difficulties connected with that problem are mainly done by the main features of the turbulent flow [9]:

- it seems a completely random and chaotic motion without no possibilities of repetition and this aspect suggests that one way in which turbulence can be analysed is through a statistical method;
- the phenomena of diffusion, dissipation and chemical mixture problems are enhanced by irregularities of turbulence that allows to different species of fluid to be mixed together increasing the exchange of physical properties among them in a faster way;

- the change of the variables of interest takes place in a very long range of characteristic length and time scale;
- turbulence must be analysed as a complex three-dimensional problem because once again the irregularities of the problem makes each variable not only a function of time and of one coordinate as in the case of the analysis of the averaged flow, but it becomes relevant also the precise position in the space of all the particles if a solution to the problem must be found.

For these motivations it is not so easy to define not only a numerical approximation of a turbulent flow, but also a mathematical model that with some assumptions is able to describe a possible solution to this strange effect present in many physical phenomena.

1.2.1 Energy cascade theory

This was one of the first theoretical schemes that was presented by Richardson in 1922. It is not connected with the numerical solution but it is a quite good theory that explains how it is possible to find a connection with the new schemes of solution for turbulence and the equations of NSF. The main principle of the cascade energy says that the kinetic energy enters the turbulence at the largest scales of motion and it is transferred to smaller and smaller scales until it is dissipated by viscous actions ([9, 10]). During the development of the theory, Richardson explained that vortexes with huge dimensions are really unstable and they break themselves in vortexes with smaller dimensions. This principle of continuous breakage and transferring of energy continue until the viscous forces are not enough strong to dissipate completely the kinetic energy compared on the small vortexes and this is the principle of the energy cascade. From the mathematical point of view, Richardson, with the support also of the Kolmogorov hypothesis, was able to develop a mathematical model able to describe the behaviour of how this transferring of kinetic energy works [9]. The first step was the description of the main parameters that characterise the biggest vortexes in a flux. In this case what is known is that the Reynold number is defined as follow:

$$Re = \frac{uL}{\nu} \tag{1.14}$$

This is exactly the same formulation as before but instead of defining the coefficient with the dynamic viscosity μ measured in $[Pa \cdot s]$, it is preferred to use the cinematic viscosity which is simply the dynamic viscosity divided by the density obtaining a quantity with a unit of measure of m^2/s and it is defined with that symbol ν. In order now to define the range in which the dissipation takes place, it is necessary to pass through Re coefficient because it is able to describe the behaviour of the inertial part over the viscous one. Moreover, Richardson defined that the large eddies have a characteristic length l_0 comparable to the l of the flux and a characteristic speed comparable with the speed of the flux itself $\mathrm{u} = \mathrm{u}_0$. So, in the end the Re number will be comparable with the number of the flux itself and it is defined as follow ([9, 11]):

$$Re_0 = \frac{u_0 \eta_0}{\nu} \tag{1.15}$$

Moreover, it is possible to define the decrease of kinetic energy density per unit mass as:

$$\epsilon = 2\nu S_{ij} S_{ij} \tag{1.16}$$

Where:

$$S_{ij} = \frac{1}{2}\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right) \tag{1.17}$$

The quantity S depends on the gradient of the velocity. This definition of ϵ is just adopted to describe what is called the Taylor microscale and it is a just a transition between the energy containing range so the large scales and the dissipation range at smaller scales also called as the Kolmogorov one as said in [9]. For this reason it is preferred to focus also in this case only on these two ranges that are more effective for the kind of analysis that must be performed instead of providing also all the possible relations in the inertial subrange describing the eddy size also in this small range. Before to go further in the mathematical formulation of the principle of how is it possible that the energy is dissipated by the vortexes at the small scale and how instead the energy is bring by the ones at the larger scales, it is necessary to describe the assumptions provided by Kolmogorov to this theory that allows to get final results together with the Richardson principle:

- 1st Kolmogorov hypothesis: there is the possibility to define a certain characteristic length under which the turbulent motions are isotropic while above that characteristic length they are anisotropic. The characteristic length of limit is defined with l_{EI} and it is imposed to $l_0/6$;
- 2nd Kolmogorov hypothesis or 1st similarity hypothesis: if the Re number is high enough, it is possible to say that the small-scale eddies are completely governed by the viscosity and by dissipation. Also, in this case there is a threshold on the length scale which instead is defined as η;
- 3rd Kolmogorov hypothesis or 2nd similarity hypothesis: at the intermediate range so when $\eta < l < l_0$, the effects of the viscosity are negligible and only the turbulent kinetic energy is relevant. So in the case of $l < l_{EI}$ if the two processes are in equilibrium each other that are the energy transfer rate τ_{EI} from the larger to the smaller scale and the turbulent energy dissipation ϵ. This is why this part is defined as "universal equilibrium range" and it is true the equilibrium:

$$\epsilon \simeq \tau_{EI} \tag{1.18}$$

Under these conditions, it is possible to say that through the two parameters ϵ and ν and from the 1st similarity hypothesis $Re \cong 1$ because in the case of small scales what is known is that the dissipation occurs caused by a very small Reynold number and viscosity effect takes place. it is possible to say that the new scales of length, velocity, and time:

$$\eta = \left(\frac{\nu^3}{\epsilon}\right)^{\frac{1}{4}} \tag{1.19}$$

$$u = (\nu\epsilon)^{\frac{1}{4}} \tag{1.20}$$

$$\tau = \left(\frac{\nu}{\epsilon}\right)^{\frac{1}{2}} \tag{1.21}$$

After the definition phase of the Kolmogorov scales or the scales in the small range written in the previous three formulations, it is necessary to make a connection with the large scales passing through the 2nd similarity hypothesis. It is possible to obtain them just making a ratio between the small scale and the large scale dividing the previous three equations by the characteristic length, speed and time of the large eddies obtaining the following equations.

$$\frac{\eta}{l_0} = (\frac{\nu^3}{\epsilon})^{\frac{1}{4}}\frac{1}{l_0} = \frac{\nu^{\frac{3}{4}}}{\nu^{\frac{1}{4}}l_0} \simeq Re_0^{-3/4} \tag{1.22}$$

$$\frac{u}{u_0} \simeq Re_0^{-1/4} \tag{1.23}$$

$$\frac{\tau}{\tau_0} = Re_0^{-1/2} \tag{1.24}$$

The following three methods instead are better adopted in order to find a clear solution from the numerical point of view of the NSF set of equations. The model proposed by Richardson and Kolmogorov was just a first approximation of how deal with turbulence in order to define a possible mathematical way able to predict the behaviour of it basing on simple parameters connected with the problem. During the years, with the development of numerical approximation methods, problems connected with the difficulty of the computation of the behaviour of a turbulent flows disappear gradually. In fact, the three methods showed below are just three ways in which the numerical algorithm deal turbulence making computations of the simple equations embedded in the NSF set of equations and at each time step making some simplifications depending on the kind of resolution required by the simulation. Anyway, there are some good proofs that with the new research the model proposed at the beginning from Kolmogorov and Richardson is good ([13, 14]). In particular, the mathematical model described in this paragraph is the one at the base of the Direct Numerical Simulation (DNS) method furtherly described because it requires the principles under the theory of the small eddies to obtain the results [15] while for the other two numerical methods present below this will not so relevant.

In the end a possible scheme of how the eddies and vortexes are split can be presented below where just depending on the type of dimension of the eddies, a different phenomenon takes place because at large scales the energy is high and the eddies are split till they reach the Kolmogorov scale at which the viscosity of the fluid is able to dissipate completely them energy smoothing the turbulence effects.

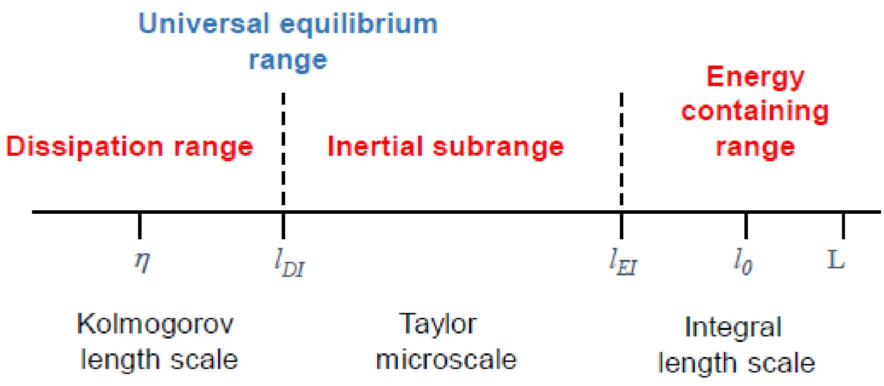

Figure 1.4: Different scales connected with Kolmogorov-Richardson theory.

1.2.2 Numerical methods for turbulence

The previous chapter describes just a possible way in which turbulence can be considered analysing how is it possible that energy is dissipated. Here instead, the most interesting part is considered because not only the effects of turbulence are analysed but it is of interest to define also how turbulence can affect the flow of the fluid so the evolution of the turbulent flow. Three different methods are presented and they are the DNS (Direct Numerical Simulation), RANS (Reynold Averaged Navier-Stokes) and LES (Large Eddies Simulation).

DNS is the most complicated method in terms of computational costs but also from this point of view the best in terms of the precision of the model. This method is very precise because it takes simply the NSF set of equations and solve them without any kind of limitation about the spatial and temporal scales being able to solve a turbulent flow up to the Kolmogorov scales ([17, 18] describe better the precision of this method) that are the one described in the previous sections in which the turbulent kinetic energy is completely dissipated by the viscous effect. For this reason, it is considered as a sort of real simulation of what can happen in the experimental model if the simulation parameters are set as much as possible closed to the reality.

The DNS method shares with the physical model of the turbulence the fact that both require a fine three-dimensional grid. Due by the high instability of the turbulence it is really clear that when it cannot be studied in a simplified model but if a quite good explanation of the effects of turbulence are required, this method is the best among the three in terms of description of the turbulent flow. The main problems connected with this type of simulation are connected with the fact that in DNS there is a particular way in which the viscosity affects the turbulence and in addition the time to get the results is for sure a real critical problem even if in the lasts years some techniques to improve this weakness are developed. For this reason it is not possible to use this technique with Re numbers that are very high but in general it is adopted when the transition phase is present because for instance it is required to analyse in a precise way how eddies are creating and evolving in time [18].
This model is strictly connected with the Kolmogorov and Richardson theory. In fact, the cascade theory says that largest eddies are the most energetic one and the geometry and the size of them is connected with the main flow of the fluid and not from the viscosity of the fluid while in the case of the smallest eddies the viscosity effect is really important because also it describes the dissipation of the energy principle. For this reason this is why the DNS is used: it must be able to have a domain large enough to collect all the large eddies in the flow and also the grid must be fine enough in order to solve the dissipation scales as mentioned in [7].

The second instead is the most approximative one and it is called as RANS. The main attention here is focused on the mean flow properties instead of considering the single eddies which characterise the flow. In this case, it is no more relevant the definition of the property analysed in each time instant because instead of take into account the exact quantity such as the speed or the temperature (depending if the momentum or the energy equation is considered), it is possible to split it in two different parts [21] which are the mean quantity and what is called as fluctuation.

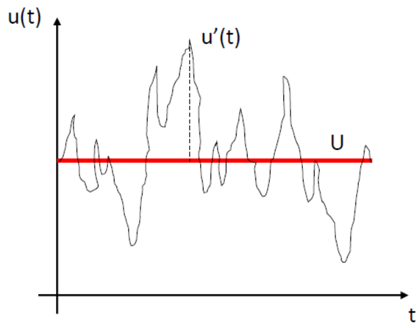

Figure 1.5: Longitudinal speed considered with RANS approach.

$$u(t) = U + u'(t), T(t) = T + T'(t) \tag{1.25}$$

In the previous equation both the speed and the temperature are written because the energy and the momentum equations are of interest. As can be shown in the figure the two parameters can be defined through a constant value of them which represents the average while the second term is connected with the fluctuations and they are instead linked to a time dependent quantity [19, 21]. The term in the capital letters describes the mean flow and it is defined as:

$$U = \frac{1}{\tau}\int_0^\tau u(t)dt \equiv \bar{u}\,, T = \frac{1}{\tau}\int_0^\tau T(t)dt \equiv \bar{T} \tag{1.26}$$

This represents the average flow evolution during time and this quantity is used as reference for the definition of the properties in a precise time instant through the local connection that at each time step is added and which is called fluctuation. The main property that must be highlighted before the explanation of how this theory can be used in the NSF set of equations is that the averaged value of the fluctuation is equal to zero.

Once defined this property, it is possible to substitute this definition in the three main equations that are continuity, momentum and energy equations. As suggested before, making the substitution between the general value of the speed with the new averaged one in the continuity equation, a not relevant thing is obtained because of the property of the fluctuating component. Instead, if the new definitions of the temperature and of the speed are introduced inside of the momentum and energy equations and averaging everything on both sides, it is possible to find that a new additional term is found in both equations [21].

$$\frac{\partial \rho U}{\partial t} = -\nabla \cdot \rho UU - \nabla P - \nabla \cdot (\mu \nabla U - \rho \overline{u'u'}) \tag{1.27}$$

$$\frac{\partial (\rho c_p T)}{\partial t} + \nabla \cdot \rho c_p UT = \nabla \cdot (k \nabla T - \rho \overline{u' c_p t'}) + S \tag{1.28}$$

The two terms represent additional unknown terms that must be solved where in the momentum equation the new terms are 6 due by the fact that it is a tensor also called as Reynold stress tensor and on the energy equation the term is characterised by a vector and for this reason the new unknowns are 3. This problem it is also called as "closure problem" because of the addition of these new terms. In order to close this problem, just a brief explanation of how it can be solved is presented below.

- At the base of the different methods, the Boussinesq approximation is a very good theory that can be used to find the remaining equations and close the problem. For this reason, Boussinesq said that the transfer phase of the momentum is caused by turbulent eddies and it can be modelled with eddy or turbulent viscosity [23]. This assumption of this scientist was focused on the fact that as for the laminar flow, there should be a connection between the turbulent flow and the gradients that created this turbulence. For this reason, the first way in which can be defined the Reynold stress tensor is due to this connection between the gradient of the average flow and the turbulent stress tensor through a new viscosity coefficient μ_t. So, the definition of the new stress tensor becomes:

$$\tau_{Re_{ij}} = \mu_t \left(\frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right) - \frac{2}{3} \rho k \delta_{ij} \tag{1.29}$$

Where the second part of the previous equation is present to avoid that the normal stresses are not cancelled. The same can be done to compute the additional term in the energy equation. Instead of having an eddy viscosity, there is an eddy diffusivity and the additional vector is computed as:

$$-\overline{u_i' t'} = \alpha_t \frac{\partial T}{\partial x_i} \tag{1.30}$$

In order to define these two new parameters, it is possible to consider the model of the mixing length which allows to define the eddy viscosity and after that obtain also the eddy diffusivity considering that both the momentum and the energy transport are due by the same mechanism. It is also expected that they are comparable in order to respect the stability condition imposed by the Prandtl number that must be almost equal to 1 that is computed as the ratio between the eddy viscosity and the eddy diffusivity. For this reason, what is called as Prandtl mixing length can be used to define the eddy viscosity [24] and it describes the length after which the turbulent flow is destroyed by the viscosity. The eddy viscosity can be defined as:

$$\mu_t = \rho l_m^2 \left|\frac{\partial U}{\partial y}\right| \tag{1.31}$$

Values of the mixing length are present inside of some tables obtained experimentally [24] and they allow to find that coefficient basing on physical parameters selected during the set-up phase of the experiment.

- the second famous model is the k-epsilon one. In this case it is proposed to compute the Reynold stress tensor adding two new equations to the previous one obtained in the case of turbulent flow. While in the case of the Boussinesq approximation no other equations were required, in this case it is necessary to add two new equations to the previous NSF set of equations for a turbulent flow in order to get the final result. In this case it is just presented a very brief introduction of the model. In fact, in this case the coefficients of eddy viscosity and eddy diffusivity are computed through two new coefficients that k are and ϵ [22]. It is necessary to link the characteristic velocity of the largest eddy with the turbulent kinetic energy k:

$$u_0 = k^{1/2} \tag{1.32}$$

Moreover, the rate of dissipation of the turbulent kinetic energy is defined just deriving the transport equation for the turbulent kinetic energy obtaining:

$$\epsilon = 2\mu s'\acute{:} s' \quad (1.33)$$

Then it is possible to define once again the eddy viscosity and the eddy diffusivity as a combination of these two new terms:

$$\mu_t = c_M \frac{k^2}{\varepsilon} \quad (1.34)$$

$$\alpha_t = c_T \frac{k^2}{\varepsilon} \quad (1.35)$$

Where c_M and c_T can be computed substituting in the momentum and in the energy equations and solving two standard transport equations.

So, in general there are many techniques that can be used in order to solve the turbulence depending on the type of physical problem that must be solved.

LES technique is the last possibility that can be adopted and it is something in between of the previous two solutions and it is, many times, also the method preferred. In the case of DNS, the solution is proposed in a very precise way and the time required is too long. In the second case instead, the method is focused on the use of the mean flow and for this reason it is no more precise as in the previous case. Finally, The LES method it is preferred not only because it is the method used also by the Fire Dynamic Simulation software, but also because it provides quite good results without needing very long times. So, it is preferred many times because it gives the proper comparison in terms of time and precision of the results.
In general, LES technique is characterised by a subdivision between the larger scales and the smaller ones [25]. The two different scales are divided through a filtering system because the larger scales are solved directly with the NSF set of equation while the smallest ones are modelled according to the larger one. For sure in this case the result will not be precise as in the case of DNS but also not so unprecise as in the case of RANS. Moreover, the time also will be better than the DNS but a little bit worst if compared to the RANS.

The filtering system can be analysed in order to explain what the sub-grid or the smaller scales are. They are not solved but they are obtained accordingly to the larger one. The fire dynamic software uses a technique in which the separation unit used in order to distinguish between the two scales is just applied with a mean averaged dimension of the grid in the three dimensions. For this reason, the solution adopted is based on the grid selected by the programmer during the definition phase of the code.

$$\Delta = \delta x \delta y \delta z^{1/3} \tag{1.36}$$

This measure is used as a low-pass filter because with this mechanism it is possible to define the averaged properties of each cell. FDS is based on this LES technique and for this reason it is better to create a grid with equal dimensions in the three coordinates. So if the problem that is analysed is studied in the three dimensions, it is better to use the same grid length on all the dimensions because otherwise they are substituted by this filter Δ and the calculations are made basing on that parameter. Once defined this filter width which is a cube with edge of Δ, any continuous field Φ it is defined following the definition present in [7]:

$$\overline{\Phi}(x,y,z,t) = \frac{1}{V_c}\int_{x-\delta x/2}^{x+\delta x/2}\int_{y-\delta y/2}^{y+\delta y/2}\int_{z-\delta z/2}^{z+\delta z/2}\Phi(x',y',z',t)dx'dy'dz' \tag{1.37}$$

This definition can be applied to the NSF system of equations used for the solution of the DNS. What is obtained after the application of this filter is a signal that is a little bit smoother. It is different with respect the averaged signal used in the case of RANS technique. Moreover, what can be noticed is that the larger is Δ so the grid space, the smoother will be the result. Otherwise if Δ decreases, this effect is reduced and if the grid is small enough, the solution obtained after the filtration system can be analysed also as a complete DNS solution. The Smagorinsky-Lilly model is the most used one and it define the eddy viscosity as follow:

$$\mu_t = \acute{\rho}(C_s\Delta)^2(2S_{ij}S_{ij} + G)^{1/2} \tag{1.38}$$

In this way, the turbulent momentum and energy equations are solved with this filtering technique at the base where the parameters necessary to close the problem of the NSF set of equations are connected to the parameter Δ.

A comparison among the three methods is presented below and as can be seen once again, it is possible to analyse that in the first case there is a very precise definition of the turbulence flow while in the last one instead of focusing on each single eddy, it is preferred to analyse the averaged behaviour of the main flow. In the middle there is just the intermediate solution that is the preferred one because of the time required to solve the NSF set of equations growth extremely from the right to the left because of the precision of the model adopted.

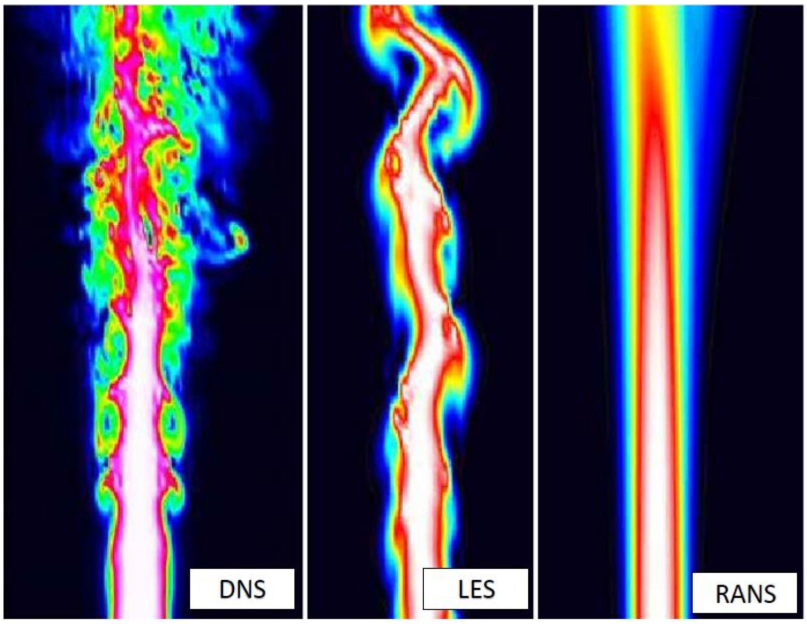

Figure 1.6: Comparison of the results obtained with the three numerical methods from a generic fluid motion.

In the lasts years new algorithms with a mixed code among the three solutions are developed [21,26] and they try to take the positive aspects of all the them in order to improve the results obtained by the numerical simulations without increasing excessively the time to perform the simulation.

2. Safety in road and rail tunnels

The safety restrictions developed during the years and present nowadays are reached unfortunately studying previous dramatic accidents in tunnels in which more than 200 people lost the life in the last century. The aim of this small overview is just to explain how it is obtained the level of safety that is present today in tunnels and also why some standards are present.

The history presents many cases in which the evolution of a fire in a tunnel becomes uncontrollable and also fatal for many persons [27]. The first that can be mentioned is another famous tunnel between France and Italy. The Mont Blanc tunnel, in 1999 was the first catastrophic event connected with fire in a tunnel [29]. The number of casualties was of 39 that was one of the highest number of victims recorded. It was necessary to work for 53 hours before to extinguish the fire and more than one week in order to decrease the temperature again to 30°C again. During the fire evolution, the temperature reaches more than 1000°C and for this reason it was impossible for the fireman to reach the source of the fire for many hours. The fire started from a truck that suddenly involves many other vehicles around of it. This is just an example that is explained in a deeper way because it is really important this analysis in the case of tunnel.

Figure 2.1: Picture of Mont Blanc tunnel after the terrible accident in 1999.

Other relevant examples were in different part of the world with also more dramatic evolutions such as the Salang tunnel where the number of deaths was equal to 172. Also, the Frejus tunnel was connected with some accidents of this type because for instance in 1993 when one HGV burns or also when in 2005 a more critical situation was created. In the second accident of the Frejus tunnel, a truck filled by tyres, after losing some fuel, started to burn and at the end the number of deaths was of 2 and of intoxicated equal to 21. A brief scheme of the most relevant and dangerous accidents happened connected to fire in tunnels is presented in the table below [27].

Tunnel name	Length [km]	Year	Time [h]	Death	Vehicles
Frejus	12,868	2005	6	2	9 cars
St. Gotthard	16,3	2001	24	11	10 cars - 13 trucks
Gleinalm	8,3	2001	0,62	5	2 cars
Tauern	6,4	1999	14	12	24 cars - 16 trucks
Mont Blanche	11,6	1999	53	39	32 cars - 2 trucks
Isola delle femmine	0,148	1996	2	5	18 cars - 1 truck - 1 bus
Pfander	6,719	1995	1	3	3 cars
Huguenot	3,914	1994	1	1	1 car
Serra Ripoli	0,442	1993	2,5	4	16 cars

Table 2.1: Brief overview on the most critical accidents of the last 30 years in road tunnels.

As can be seen the number of accidents is very high just in these few years and it becomes larger and larger if older events are considered connected also with the fact that the level of safety was not high as today. One way that can increase really the level of safety in tunnels is also connected with the new way in which numerical simulations are implemented because it is not so easy sometimes to make all the possible tests directly on the final product, especially if the final component is a long tunnel.

2.1 Brief overview on standards development

Safety in tunnel is a really complex task with respect the common road applications. The problems connected with safety in the long tunnel created in the last century start to move people and mainly engineering in order to define some safety features and avoid catastrophises as in the Mont Blanc for instance. In fact, Mont Blanc and Tauern accidents move the attention to these kind of problems in order to try as much as possible to avoid further things like that. PIARC or "*Permanent International Association of Road Congress*" that was founded in 1909 [28] was the first association that was focused on this task because starting from the 1995 it began an analysis on the transportation of dangerous goods though road tunnels in according with OECD which is the *"Organisation for Economic cooperation and development"*. In the same years when the Mont Blanc accident happened, Switzerland, France, Austria and Italy created an informal group called as Alpine Countries group named by the WERD or *"Western European Road Directors"* and in the end they approved the new safety conditions imposed by this informal group. The most important document written about tunnel safety and all the conditions that an accident can create is related with the *"recommendations of the group of experts on safety of road tunnel"* which is published in 2001 by once again PIARC that is the maximum expert about this subject. This document was confirmed in 2004 when EU Directive published one document called *"directive 2004/54/EC of the European parliament and of the council of minimum safety requirements for tunnels in the trans-European road network"*. The aim of this document was first to impose a minimum level of safety on the European tunnels trying to minimise tragedies such as the Mont Blanc one and making sure that all the possible ways in which an accident can be controlled are respected by the tunnel stakeholders. For instance one important point to guarantee a good level of safety is imposed by the structural support that must be able to maintain the structure of the tunnel even in case of fire and the reason is connected once again with the life safety because in the case of a huge failure of the tunnel, it can be a possible flooding of it and even if the damage is far away from the rescue services for instance, it can cause additional victims also after some days from the accident. The second aim that this document want to highlight is that once all the standards focused on the life safety are respected, it is required to consider also other aspects such as try to minimise the damages on the structure and take care about the economic and environmental conditions. These conclusions are connected with the fact that for all the accidents mentioned in the previous chapter, there are strong repercussions in time and also very huge amount of money required to normalize once again the situation.

At the end of all these explanations it is required to have a look also in terms of the economic damage. The economy here is not only connected with the real costs of the reconstruction of the tunnel but also with the implications that this damage has on the society [28]. One clear example can be once again the Mont Blanc catastrophe. In this case in fact, the expenses connected with the reconstruction of the tunnel were estimated to a value of 250 million of euros but with a socioeconomic analysis this amount rises up till the value of 1.75 billion. This can be connected with the fact that the passage can damage the transportation among the countries of some products that for sure will be moved with different ways and for sure more expensive.

2.2 Safety in road tunnels

Tunnels are fundamental for the transportation of the modern countries. They allow to decrease the transportation time for both road and rail vehicles. The effects of the increase of the traffic all around the world make tunnels essentials for the communications among different countries.

For sure the situation is a little bit trickier when an accident happens in a tunnel because it is not so easy to preserve people who are inside of it as in case of fire in an open environment for instance. Moreover, it is also required that from the structural point of view, the tunnel must be designed in order to be able to support high thermal loads and in order not to fail under firing conditions. Due by the increase on the number of vehicles that use tunnels during all days, it is necessary to verify the conditions of them under different aspects. The construction phase becomes much more complicated for different factors:

- the infrastructures created through mountains all around the world become very complex and also in length. For this reason, the safety conditions for a very long tunnel are not so easy as in the case of old short and urban ones;
- New materials are transported and new materials also really dangerous that can explode creating not only problems directly connected with the

explosion, but also the possibility to destroy completely the entire structure;

- bidirectional tunnels are coming out in the last years and also new types of tunnels are created mainly in the urban environment where cars, trains and buses have to share the same infrastructure and it for sure increases the difficulties in the prevention phase of accidents and also in the controlling phase of them;
- mechanical defects and imperfections in the machines that travel through the tunnel where many times are the cause of very dramatic accidents as in the case of Mont Blanc.

Figure 2.2: Example of the complexity of Blanka tunnel in Praga.

For these reasons as PIARC document [28] suggests, at the base of everything during the designing phase there should be the prevention. The first most important thing that can avoid any major problem is the prevention of it. When it is not possible to prevent some accidents instead, it is necessary to consider how solve and reduce the effects of this making possible the condition in which:

- people involved directly in the accident must be able to rescue themselves in some safety regions defined in the designing phase;

- sudden intervention in order to limit the consequences. It is well known that the fire evolution is not extremely rapid but some time is required to reach the maximum of the fire and as a consequence the worst conditions to extinguish it;
- ensuring different ways in which the emergency services can help people involved in the fire such as safety tunnels or emergency lines;
- protect the environment because as said before a critical damage in the structure can create in addition to a real increase in the amount of money to repair the damages, also a consequence for more dangerous situations;
- control the material damage for the same reason explained in the previous point.

As a consequence, the graph below embeds all the elements that can affect the safety conditions in the tunnel and they are briefly explained in the following steps.

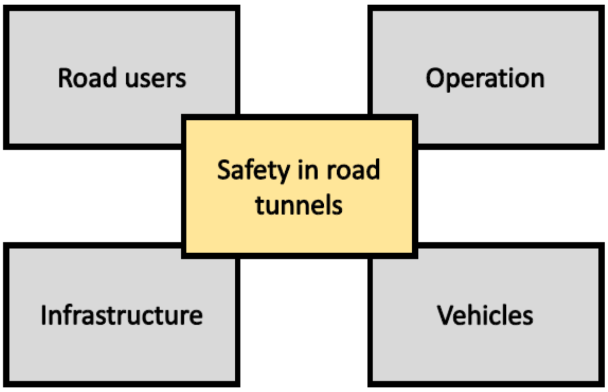

Figure 2.3: Scheme of factors that affect safety in road tunnels.

In this section it is not a detailed description of all of these four factors that can influence the safety conditions but there is simply an explanation of the general behaviours and from a generic point of view of what can create dangerous conditions in tunnel touching all the four points.

Road users

As mentioned in PIARC document, one of the most important point when the causes for an incident are analysed is connected with the presence of human errors.

These unfortunately can never be corrected or eliminated at all due by the fact that they are unexpectable. The only thing that can be done is once again work with the prevention as much as possible and also try to limit the consequences of them. For these reasons, some general rules about the correct behaviour in tunnels are suggested in the document and they are mainly focused on the prevention of possible incidents caused by human mistakes. A quick list of them is presented below.

- Information campaigns: every user of the tunnel must know the basic correct behaviour that must be kept in tunnels.
- Driving tests: it means that each driver has to know how to deal as in the case of breakdown, congestion and accident case.
- Drive out burning vehicles because it can be easier for the rescue teams work with a situation out of tunnels an also a less dangerous condition is created inside of it.
- Roadside checks: a continuous control phase must be applied through different systems such as X-ray in order to check if some strange materials are transported by some vehicles.
- Test for professional drivers are done periodically and more frequently because the risk when the vehicle is a truck or a bus is higher.
- Test and regulations for dangerous good drivers: according also with the European law, it is imposed that the transport of dangerous products must be controlled in a deeper way and also it is necessary to declare if some of the goods transported could be dangerous or not. The law defines five classes of dangerous vehicles and depending on which class the good transported is, there are different restrictions and rules about how transport them.
- Overtaking, distance among vehicles and speed limit are instead imposed in order to prevent accidents once again or to limit them effects.

Operation

The operations necessary for the correct functioning of everything inside of the tunnel are different. Depending on the task that must be accomplished, different kinds of skilled persons are required.

- Operator: essential for the prevention phase of possible accidents because one of the aims is to maintain in a proper way all the service plants such as the ventilation system, the lighting and the traffic control systems. Moreover, they are strictly prepared on what to do in case of accidents.
- Traffic police: they have to monitor the traffic conditions and mainly it is necessary them intervention in case of accidents organizing the evacuation of tunnel and avoiding as much as possible traffic congestion.
- Emergency services: strictly prepared to worst conditions of fire or other dangerous situations. They have a precise preparation on how work in that cases.

Another important aspect is a plan in case of catastrophic act. In the emergency response plans, there is an explanation of what are the steps that must be followed in that case. The first thing that must be done is the reporting phase of the alarm to the rescue teams and each of the rescue teams know what to do. Then an automatic or a manual programme of how control the traffic is delivered trying to let out vehicles as much as possible. Each tunnel has its own emergency plan depending on the type of services provided to the tunnel itself.

Infrastructures

This is the most relevant part connected with the designing phase because a huge part of the safety environment that is created in tunnels is connected with the infrastructures. There are many elements that must be considered and coordinated in order to guarantee the proper level of safety even in tragical situations. With infrastructure it is considered everything about the apparatus different from the structure itself but necessary for the proper working condition of the system.

- Structural components: it means that all the structural elements present without considering the pipes or cables must be designed taking into account the worst fire condition in order to be safer as much as possible.
- Ventilation: is the most important system that must be considered in order to prevent and limit the effects of a fire. With the evolution of the vehicles, the fire condition becomes the main problem that ventilation system has to

accomplish because the problems connected with the normal emissions of vehicles are decreased year by year until they become quite irrelevant as in the case of electric vehicles for instance. Many kinds of ventilation systems are present depending on the kind of tunnel that is considered. These systems also explained in this part, will be better treated in the following sections.

- Other electromechanical equipment: such as for instance the sensors spread along the tunnels, the systems for issuing warnings and instructions to the road users or equipment for eliminating hazards.

Vehicles

In the lasts years the new vehicles become really less dangerous if connected with the safety in tunnels. The quick development that automotive research made in the last 10/15 years is incredibly fast and for this reason less critical motivations connected with vehicle problems can be founded. The restrictions imposed by the law allow this quick evolution in the safety conditions of the vehicles. In the past many problems were connected with the braking system that becomes really hot if damaged by the usage or connected with the electric parts of the cars that in case of short circuit automatically switch off the electric system making the vehicle safe also in these cases. Some additional restrictions are added instead with trucks or in general with HGV vehicles because in this case the problems can be of different types and also stronger in the effects. For instance some restrictions that are imposed on these vehicles is that if the products that they are transporting are really dangerous, it is present a restriction on the quantity of fuel carried by them in order to diminish the effects of a possible accident. Materials on which vehicles are made must be not inflammable and should be able to increase the calorific capacity of HGV but also of common cars.

3. Ventilation systems

Before to go further with the definition of the tunnel analysis for this work, it is necessary to make a brief introduction on the main ventilation systems present in the different tunnels all around the world. From a generic point of view, basing on decree 264/06, for tunnels longer than 1000 m is required an analysis on the quality of air and successfully a proper ventilation system able to accomplish to these three tasks can be selected:

- take under control the quality of the air under normal traffic conditions. There are different restrictions on the quality of the air inside due by the fact that internal combustion engines produce toxic gases that could be really dangerous if the percentage of them increases too much;
- to control the pollutants also during the construction phase of the tunnel itself in order to make a safe situation also for workers;
- Control the amount of smoke and try to limit the effects of a possible accident inside of the tunnel.

In general, it is not so easy to decide which kind of ventilation system adopt because different methods are available today. Some parameters can be used in order to select the proper ventilation strategy:

- the length and the geometrical construction parameters of the tunnel analysed because many times some ventilation strategies are not possible at all basing simply on the geometry of the tunnel;
- the location on which the tunnel is placed;
- the environmental and the climatic conditions;
- the density and the frequency of the vehicles inside of the tunnel.

As will be better explained in the following chapters, there are different parameters that must be considered when a ventilation system is analysed. For sure the emissions of the vehicle are the main ones because under normal conditions it is required that people who has to travel without vehicles inside of the tunnel can do it under safe conditions. For this reason, some limitations on the amount of CO and

more noxious NO_2 are imposed. These gases if present above certain limits can be really dangerous for human life. For instance, nitrogen dioxides present over a threshold 1 ppm cannot be supported by human body. In case of carbon monoxide instead, the limit is a little bit higher because in general it presents less critical effects.
Finally, the last consideration that must be analysed is also the visibility that could be affected by not only the exhaust gases produced by internal combustion engines but also by the particles produced from the tyre in contact with the road. For this reason tunnels where the truck traffic is more relevant than passenger cars will have a higher influence from the point of view of visibility because depending on the inertia of the vehicle for sure a higher consumption of tyres will be present and also more particles will be lift in the internal atmosphere of the tunnel decreasing the visibility. Another aspect that must be considered is connected with the directionality of the tunnels because if the traffic is congested in two ways, this will be for sure a factor that decrease once again the visibility and the ventilation system must consider also this point. All these passages will be better explained in Appendix A [30].

All the considerations that are made at the beginning of this explanation are made just in order to highlight the difficulties during the decision of the ventilation system. In addition, the ventilation system becomes of fundamental importance in case of fires because it is one of the most important systems that can be regulated in order to control and limit the effects of the fire. In the last years, the ventilation systems become more sophisticated since control devices are added to the fans making them the possibility to change according to the different situations present in the tunnel. Anemometers, thermometric probes, carbon monoxide analysers, nitrogen dioxide analysers opacimeters, traffic counters are only part of the sensors added in tunnels in order to control the ventilation devices optimising them.

Ventilation strategies adopted for tunnels could be of two main categories which are natural ventilation systems and mechanical ones. In the following sections the two methods are presented considering weakness and advantages of both.

3.1 Natural ventilation system

This is the easiest method adopted only for very short tunnels. In general, this strategy is used when the tunnel length does not overcome 1 km [32]. The natural ventilation system is strictly connected with the environmental conditions such as the pressure and the temperatures. In fact, both these two effects can create the natural motion between the portals and this effect is present not only on short tunnels but also in long ones [35]. As said before, the natural ventilation can be designed only for very short tunnels and the reason of this is done by the fact that the pollutants produced by the cars can be easily removed from the very small natural difference in pressures or temperatures but also by another physical principle which is called piston effect.

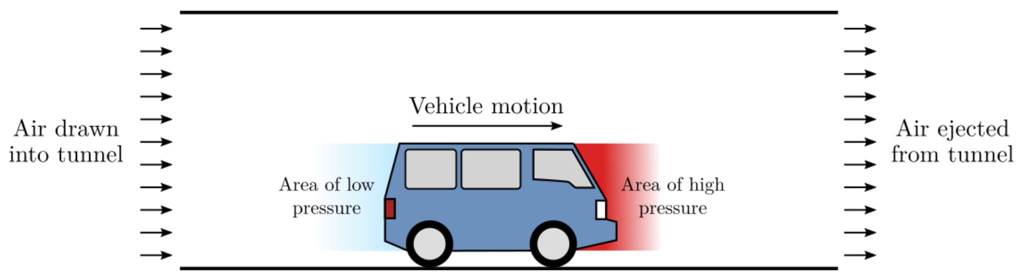

Figure 3.1: Illustration of the piston effect for natural ventilation in tunnels.

The piston effect is generated when cars travel along tunnels because of the fact that according to the Bernoulli principle, when the car passes through the tunnel, it brings the air to move due to the viscosity of the air and of the roughness of the vehicle itself. For this reason, on the back side of the car it is created an area in which the speed is increased and (as a consequence of the Bernoulli principle) the pressure is decreased. This creates a natural motion of the air from areas in which the pressure is higher to areas in which instead it is lower.

All these aspects cannot be neglected when long tunnels are instead analysed. In fact one of the effects that will be shown in the Frejus tunnel analysis in case of fire is connected with the fact that at the two portals there is always a relevant pressure difference [31] between them and for this reason, depending on the area in which the fire is located, the reaction of the ventilation system will be different because of this natural wind generate through the tunnel that can help in some cases [36], but it could be a huge problem in other problems.

In general the natural ventilation system is preferred [34] because as the name suggest, nothing must be added to the tunnel and for this reason it is not required an increase in costs due by the price of fans, the price of electric power required to make them to work and also to the maintenance that must be done on these devices. On the other hand it should be considered that in case of accidents, it is not so easy to control and to limit the smoke propagation as it should be done increasing the risk for the tunnel users and also for the rescue teams because fans are not present to control at least the propagation of the smoke inside of the tunnel itself.

3.2 Mechanical ventilation system

In this case instead, one of the main conditions that must be considered is connected once again with the length of the tunnel. A table can be presented below which shows the connection between the tunnel length and the type of ventilation system [32].

Type of ventilation system	Length [km]
Natural ventilation	< 1
Longitudinal ventilation with jet-fans	< 4
Longitudinal ventilation with smoke extraction	< 6
Semi-transversal ventilation	> 2
Transversal ventilation	> 6

Table 3.1: Selection of ventilation strategies basing on tunnel length.

In tunnel fires, the first action that must be adopted is the prevention as suggested also in the chapter 4. When it is not possible with prevention actions avoid accidents of that type, it is possible to react against the fire propagation in two ways:

- passive protection: it embeds everything which in case of fire makes more difficult its propagation without the necessity of any activation of any plant.

Some examples of this type could be for instance a previous analysis on the material in such a way that the best fire resistance ones are selected for the proper applications limiting for this reason the amount of fuel that can fed the fire. Or also other examples could be to make the structure of the tunnel with correct structural studies being sure that in catastrophic elements the structure will not break such as also the presence of many emergency exits placed in strategic point of the tunnel;

- active protection: it instead embeds all the elements present in tunnels that must be activated by human hands or also automatically, but they require an external action to be activated. In this case for instance all fire extinguishers, all the sprinklers, all the fans that are adopted for this task make part of this category of active protection from fire.

In this case the main systems on which this research is focus is connected with the active devices and particularly to ventilation systems and how they must be applied according to the different situations. For this reason they are analysed in a deeper way considering all the possibilities that can be used and which kind of distinction must be done when it is required to take a decision from the point of view of the designing phase.

3.2.1 Longitudinal ventilation

The longitudinal ventilation system is characterised by some longitudinal fans that can be used to blow air inside of the tunnel or to expel the vitiated air from the tunnel to the environment.

Figure 3.2: Couple of jet-fans for longitudinal ventilation system in tunnel.

For sure once again the number of fans required and if some expellers are required depend on the length of the tunnel and on the conditions on which the tunnel has to work. Usually the jet-fans placed inside of the tunnels where longitudinal solution is adopted are able to work on both directions because once again the other parameter that must be considered is the location of fire. What can be done in this case to design the situation with these jet-fans is connected with the momentum that can be converted in static pressure. Through the data provided by the fan builders, it is possible to consider the presence of a fans in some locations along the tunnels as pressure drop converting the effects given by these devices (the change in the speed of the air so it means the change in the momentum of the air) in a simplified overpressure [33].

$$\Delta p_{j0} = \rho Q_j (u_j - u^*)/A_T \tag{3.1}$$

This is the theoretical equivalent increase in pressure that one fan in a tunnel can provide but it is not like that because it is necessary to consider also all the possible losses.

$$\Delta p_j = \Delta p_{j0} \eta_1 \eta_2 \eta_3 \tag{3.2}$$

Where:

- η_1: it depends on the eddies and on the vortexes created at the outlet section of the fan. In general, it is possible to prove that this reduction is almost of the 10% with respect the nominal computed value of the pressure rise expected.
- η_2: this coefficient represents instead the possible losses that can be created in proximity walls or other geometric features that can affect it from every point of view. It is a factor closed to the localised losses coefficients.
- η_3: it considers the influence of some deflectors and if the jet exiting from a fan is deflected before a certain parameter decided once again experimentally. The limit imposed is about 10 times the hydraulic diameter of the tunnel without any kind of deflection in order to have this reduction coefficient equal to one and as a consequence a higher effect of the fan

considered. There are some particular deflectors that decrease this distance to values od 6/8 times the hydraulic diameter of the tunnel changing the direction of just 5/10°. If the deviation of the jet is higher than this value, it is no more possible to consider that parameter equal to 1 but it will be less.

Every reduction of the Δp_j represents an inefficiency of the fan in the overall tunnel. This higher efficiency can be translated in a reduction in price of the ventilation systems.

Last point analysed with the longitudinal scheme is connected with the number of fans required for a specific tunnel. In order also to define the number of fans required according to the data provided by the constructor, the overall resistances in the tunnel are considered as lost in terms of pressure [33] and at the end the summation of all the losses will be used to compute the variation in pressure necessary by all the fans. The three factors considered with these overall resistances are defined as:

- piston or drag effect per traffic lane: this embeds all the possible losses in terms of pressure that each car can cause passing through the tunnel and connected with the piston effect mentioned before;
- resistances due by the tunnel walls: here what are considered are the materials on which the tunnel is made because depending on the kind of superficial roughness, also the pressure drop desired by the tunnel will increase. If for instance the tunnel is quite old, the superficial roughness of the internal surfaces is increased very much due by abrasion or corrosions [38];
- Meteorological or thermostatic counter pressure: as mentioned before, this kind of natural ventilation is always present also in very long tunnel. During the designing phase, it is considered as a negative effect in order to be safer. In fact, sometimes, depending on the position of the fire inside of the tunnel, if the fire is generated in some locations where it could be convenient try to win the natural wind generated between the two portals, it is required that the fans will be able to generate this additional counter pressure able to act against the natural ventilation.

Each of these factors is characterised by some mathematical relations and each of them is characterised by empirical or experimental coefficients and this is not the aim of this job but it is presented from a methodological point of view what are the steps used to design a longitudinal ventilation system.

3.2.2 Transversal ventilation

This is the application of interest in the following case study. More in general, these ventilation systems are adopted when the tunnels are very long. There are two ducts parallel with respect the main axis of the tunnel adopted respectively for the fresh air and for the vitiated one [31]. The new fresh air is provided through lateral small vents disposed in a regular way along all the tunnel and in general placed in the lower part of the tunnel while for the vitiated air, there are some shutters disposed along the main axis of the tunnel but with bigger dimensions and less frequently than the fresh air sources. In general, vents for the fresh air are present each 5/10 m while dampers are present every 50/100 m and they are placed on the top of the tunnel and they can be controlled automatically or manually. This kind of ventilation solution in general is adopted for bidirectional tunnels longer than 4 km [33] because the main advantage of this solution is that it is possible remove the smoke exactly where the fire is generated. This system is more efficient if the mass flow rate of smoke that can be captured by dampers is well computed and it does not allow any spreading phase of the smoke itself along the tunnel [36]. Moreover, an additional requirement by this system is that a certain mass flow rate of fresh air will come in area of the fire in order to limit the propagation of the smoke.

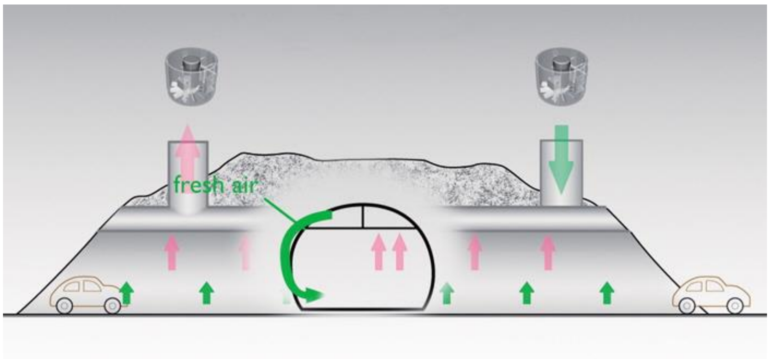

Figure 3.3: Schematic representation of transversal ventilation system.

One thing that must be considered when fire starts inside of tunnels is the time for the activation of the security systems because before that sensors start to catch something and before that dampers become effective with respect the smoke extraction procedure, some parts of the smoke will be propagated inside of tunnel and it could be dangerous. For this reason, a good level of sensitivity and a proper design of dampers is required also due by this additional time.

3.2.3 Semi-transverse ventilation

The semi-transverse model is something in between the previous two solutions. In general the flow of fresh air is provided through the portals using longitudinal fans while the ejection of the exhaust gases (or in case of fire the extraction of the smoke) is the aim of dampers disposed along the tunnel as in case of transverse ventilation. The main problem of this solution is once again connected with the stratification of smoke. As in the case of longitudinal ventilation systems, the velocity of the air provided along the tunnel is fundamental in order to control the stratification of the smoke avoiding the dangerous effects of back-layering [40].

The advantage of this solution with respect the longitudinal one is connected with the possibility to extract the smoke closer [33] to the point in which fire occurs without spreading it all along the tunnel. This is the reason why the semi-transverse ventilation is adopted when tunnels are quite long, but not too much to adopt the complete transverse solution. These solutions can be also used in different manners because it is possible to use the circuit in an opposite way with respect what is described up to now so the fresh air can be provided by the dampers on the upper part of the tunnel and the smoke can be released through the portals depending on the kind of situation analysed.

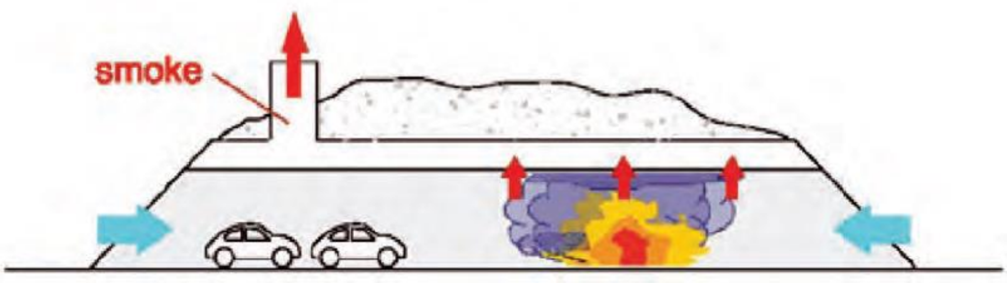

Figure 3.4: Schematic representation of semi-transverse ventilation system.

The reason why this kind of solution is adopted is because it has the huge advantage of reversibility so depending on the type of fire that is present in the tunnel, the ventilation system can be tuned depending on the different situations.

4. Preliminary analysis on Frejus tunnel

The work is focused on the analysis of a particular case that is the Frejus tunnel. The tunnel is one of the most important connecting ways between France and Italy. In particular it connects Modane in the France side with Bardonecchia in the Italian one.

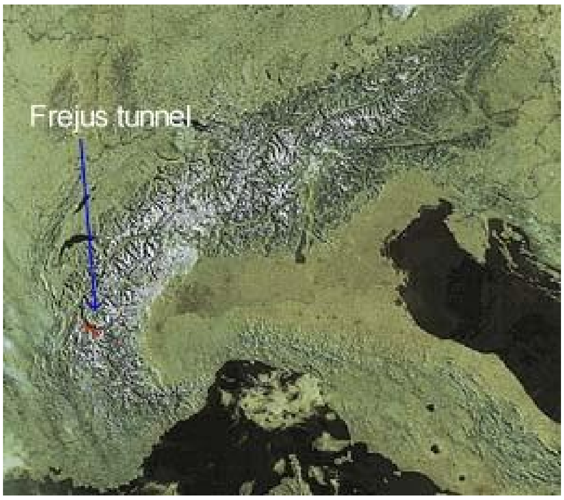

Figure 4.1: Geographical position of Frejus tunnel.

It was opened in July of 1980 and it is younger than the trail tunnel near to it. It is monitored by two societies that are the SFTRF and the SITAF and they are one French and the other Italian, respectively. Geometrically it is long 12 868 m and for this reason the two societies split it almost in the middle to manage better the tunnel [41]. In fact, 6 800 m are considered in the Italian side and the other part instead is handled by the French one. In the firsts 20 years, more than 20 million of vehicles travel through this tunnel [42]. As mentioned before, many times the standards developed during the years are unfortunately obtained basing on previous disaster. In fact, after the terrible accidents of the Mont Blanch in 1999, in 2000 the companies enrolled in the control of the Frejus tunnel, decided to improve the safety inside of it in case of fire. For these reasons new and really sensible sensors are added inside of the tunnel in order to perceive suddenly if something is wrong and stop the problem at the beginning. Moreover, new restrictions on the vehicles are imposed because it is not possible to overcome a speed of 70 km/h and it is imposed a minimum distance among the vehicles of 150 m [29]. 11 new dampers are spread along all the tunnel and a new fire hydrants system is installed putting them every 130 m [41]. Even if these new restrictions are imposed to limit terrible accidents, one the June of 2005, two truck drivers lost their life in an accident and the tunnel was closed for 2 months in order to make everything safe. After that accident, it was decided to define an additional security tunnel.

4.1 Frejus geometry and constructive aspects

Frejus tunnel is characterised by a classical "U" profile shape [36,41]. The constructive aspects are described briefly because they will be used in the following simulations. The most relevant thing is that it is a tunnel with a length of 12 868 m. This size evidences the importance of analysing the effects if fire particularly because many people can be affected by the consequences that can be generated by a possible fire inside of it. It is a bidirectional tunnel with two ways with a dimension of 3,55 m each divided by a double continuous strip. The overall dimensions of the available road are of 9 m. There are 2 sidewalks with dimensions respectively of 50 and 60 cm for an overall width of 10,10 m.

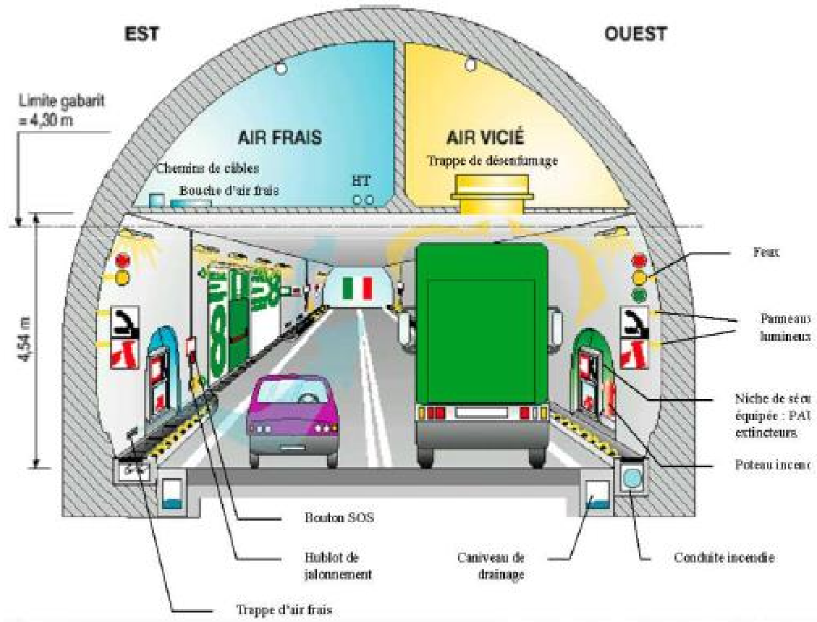

Figure 4.2: Section of Frejus considering the French portal as entrance.

Another relevant constructive factor is the difference between the height at the two portals. In fact, the Italian side present a high of 1297 m while the French one of 1228 m. This difference will cause a natural ventilation that will be discussed in the following chapter.

Connected to that there is also the available section that can be used for vehicles because due by the length of the tunnel, it is required for sure a transversal ventilation system and for this reason two ducts are required on the top as can be seen in the previous image. One duct is used in order to distribute the fresh air along all the tunnel with small vents that distribute the air taking it from the principal duct through some channels. These vents are placed on the east side with a high of 0,5 m from the floor and with an inter-axis distance of 4,5 m. The other duct instead is adopted to eliminate the vitiated air. About the available section, it is decreased really much from what expected because the ventilation system is put on top of the tunnel and it occupies a good portion of the section as shown in the previous image and as can be seen, the available height is decreased to 4,54 m. Moreover, there is another limitation imposed by the other plants attached to the ventilation ducts (such as lights, sensors, …) that decrease once again the high to a value of 4,30 m [41].

4.2 Ventilation system of Frejus road tunnel

As said before under normal conditions the system is quite easy because there is a transversal solution which allows to create a continuous change of air from the internal part of the tunnel to the external environment. In fact, under normal conditions the fresh air provided by the ventilation system is equal to the 10% of the overall capability of the ventilation system without any necessity to extract it [36]. In general it is preferred to have the mass flow rate of fresh air higher than the mass flow rate of vitiated air extracted in order to be sure that the air inside is completely removed and no huge concentrations of residuals are present in the internal atmosphere of the tunnel. Some data about how the ventilation system has to work under normal conditions are present in [41, 43]. The scheme of the ventilation system is quite easy because the air is distributed along all the tunnel through longitudinal fans that operates separately with respect the environment where vehicles are present just because it is better to inject the fresh air with a transversal solution in case of long tunnels. There are 6 ventilation stations equally distributed along the tunnel giving a subdivision of them of 2100 m [36]. Each section is characterised by a couple of fans dedicated to the injection of the fresh air and a couple of fans instead dedicated on the extraction of the exhaust gases. These 6 stations are fed depending on the area where they are. The two sections close to the portals are fed by the ventilation plant close to the respective country in which they are (section 1 by France and section 6 by Italy). Then for the intermediate 4 sections, there are two additional power plants and once again one is French and the other Italian used to provide power to the fans of the section 4 and 5 in case of the Italy and of the section 2 and 3 in case of France. These power systems are put exactly at one third of the overall length of the tunnel in order to create a certain balance in case of accidents.

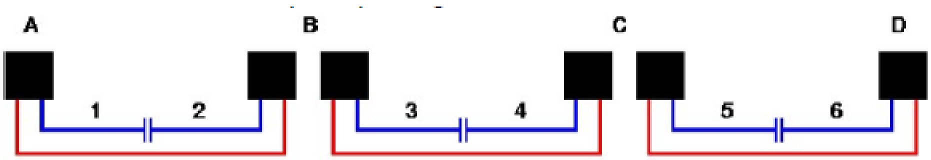

Figure 4.3: Schematic representation of the six stations of ventilation.

As can be seen, depending on the level of some relevant dangerous elements present in the tunnel, the different parts are activated with also a controlled power depending on the result desired. In general, under normal conditions, there are different sensors that are used in order to measure the amount of that noxious gases and opacity that vehicles cause. The fire is just a particular case of this common situation due by the fact that when a fire happens, for sure some smoke will be present and for sure the sensors enrolled in the evaluation of opacity inside of the tunnel will be activated. For this reason, depending on the decreased levels of visibility or depending on the huge increase in toxic particles produced, it is possible to move the fan to a point of them characteristic in which the mass flow rate becomes really high. In the extreme conditions, in case of a really dangerous fire situation, the system is designed in order to be able to provide an amount of fresh air up to 1420 m^3/s and an extraction equal to 240 m^3/s just in precise zones. In fact the system that is adopted in case of fire is not designed to extract the smoke from one of the two portals as in the case of simple longitudinal ventilation tunnels, but it is preferred to focus the smoke distribution simply in a portion of the tunnel where the fire takes place in order to make all the rest safe for the other travellers. In case of fire this localised expulsion of smoke is created through the presence of some dampers with dimensions of 10 m^2 almost and they can be controlled automatically or manually and that are distributed along the tunnel with a distance of 100-130 m.

In order to control e limit the areas in which the smoke is diffused, it is required to consider also the position of the fire and the difference in pressures that will be in that precise moment between the two portals. This will be the most relevant part connected with the analysis of fire in this tunnel.
In addition, to be sure that the area where the smoke is diffused will be limited in a restricted region, in correspondence of the two intermediate electrical plants, two chimneys with an angle of 40° and a length of 700 m with an internal division that allows to split once again the vitiated or toxic air from the fresh one. performing this kind of solution, in case of fire, it is possible to work with different operating conditions making possible to activate fans in different ways and reacting in a proper way according to the situation [44].

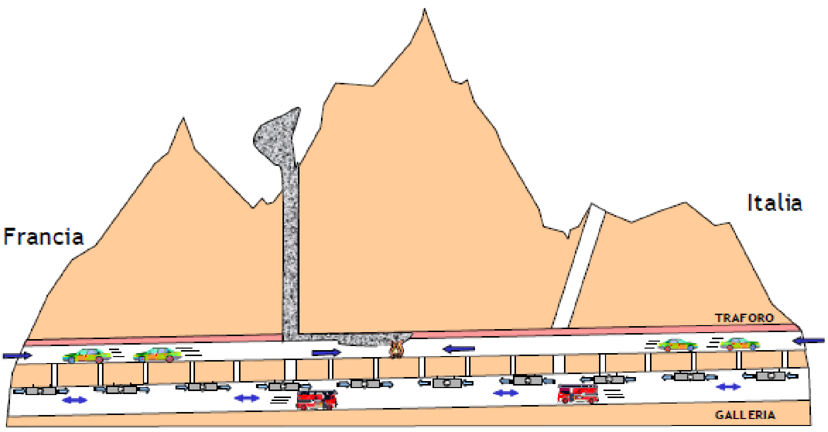

Figure 4.4: Simple representation of how the system reacts in case of fire.

4.3 Fire dynamic simulator: software adopted for simulations

Fire dynamic simulator (or easily FDS) is a software developed by the NIST (National Institute of Standards and Technology) which diffused the first version in 2000. The first step that must be done before the characterisation in detail of the simulations is connected with the description of the software adopted. The fluid-dynamic software FDS collaborates with a graphical software called Smokeview. This last one is adopted only to show graphically what FDS is able to obtain after numerical computations. From a generic point of view the software is designed to solve numerically the Navier-Stokes-Fourier set of equations described in the first chapter considering one strong limitation. In this case a huge assumption is adopted which is connected with the Mach number [7]. Experimentally can be proved that if the Mach number is lower than 0.3, the density is no more a variable but it is a fixed value connected simply with the chemical species considered. For this reason, in fact, it is possible to collect the density from all the integrals present in the set of equations reducing too much the problem and also reducing the number of calculations that should be performed. Moreover, as the name suggests, this software is focused on the study of the propagation of smoke and of the heat exchange inside of the virtual environment created. For this reason, FDS is really indicated in order to study the effects of fires and the connected consequences. During the years, the distributors of the software try to make it more sophisticated and one way to do this is to study the behaviour of the combustion in order to evaluate better the propagation of the products obtained after the combustion because many times they are the cause of death in case of accidents.

Even if the equations able to describe these kind of thermo-fluid dynamic problems are present from a century, many years were required in order to obtain a software able to describe the behaviour of fire and the evolution of the smoke at the end of the combustion process. Since the developing phase of the computers, there was this desire to apply numerical simulations also for these kinds of problems. The main limitations that stop the evolution of a software like FDS were connected to three reasons:

- the scenarios that can be created under fire conditions are too many and also really different each other;

- there were also limitations connected with the technology because also today a quite good computational power is required to perform all the calculations and it is not so easy to get the solution. In the past when the computers are not so good, even if from the theoretical point of view, it was everything listed under mathematical equations, it was impossible to implement them in a virtual environment to perform the calculations;
- finally, there were some limitations from the theoretical point of view also when combustion or more in general the fuel of the fire was not considered. In fact, only in the last years it was possible to define mathematical equations able to consider also the aspects connected with the fuels.

The development of the software brings the producers also to try different kind of situations and different mathematical techniques in order to obtain the final code used nowadays. In fact, what is explained in the Chapter 1 is something like a description from a generic point of view of the mathematical equations implemented behind the software and also the description of all the points is maintained considering the development of the software itself. As can be seen, at the beginning, the RANS method would be tried as the method used to schematise the description of turbulence. In reality as also explained before, in order to describe better the turbulent behaviour, it is convenient to consider not the DNS method but the LES one because it was proved that the efficiency in terms of time was improved really much and also the accuracy of the model is sufficient for the kind of applications adopted. The type of simulation adopted is very important because in the FDS software can be implemented both LES and DNS techniques. The main distinction between them is connected once again by the time and the precision of the solution that will be obtained. Moreover, the large eddy simulation can be also of two additional types which are the VLES or Very Large Eddy Simulation or SVLES or Simple Very Large Eddy Simulation.

The large eddy simulation approach is mainly adopted because it is able to use a low-pass filter in order to consider only the eddies and vortexes that can have relevant effects on the development of the fluid and not all of them.

4.3.1 Features of the software useful for Frejus tunnel simulations

FDS is a software based on numerical solution of the NSF set of equations as suggested before. In this case, it uses a numerical approach to close the problem and in general, numerical methods can be performed splitting the volume in many smaller ones and after that, through the use of some boundary conditions, solve the equations imposed in the model for each of the small volumes created. The main principle which describes how the software works is connected with the creation of a file where all the informations useful for the software are put and they are listed in order to make possible the solution of the problem.
Inside of this text file, the first parameter that must be imposed is connected with the mesh. In general, the mesh is just a way in which it is possible to define the behaviour of one body from the internal point of view and not only from the external sides as in the case of portals of tunnels. In fact, the creation of the mesh is simply a trick that is used in order to analyse the behaviour the internal behaviour of a system and in this case of the tunnel creating many cells that communicate each other passing informations each others. This technique can be used also for other types of applications such as the approach adopted in case of structural environments where instead of the evolution of the heat or of the smoke it is considered the distribution of stresses and tensions inside of the body considered.

As suggested before, the main parts of numerical simulations are performed basing on LES approach and this property is also the default one applied to all the simulations. For this reason, once again the definition of the mesh must follow some restrictions because it should be characterised by prismatic shapes with the same dimensions in the three directions. A suggestion of the dimensions of the grid is proposed by the software distributors. As a first guess, it is possible to impose the mesh grid using the following formulation to define the dimensions of the grid:

$$D^* = \left(\frac{\acute{Q}}{\rho_\infty c_p T_\infty \sqrt{g}}\right)^{2/5} \tag{4.1}$$

The value obtained from this relation is just an indication of the dimensions of the cubes that characterise the mesh. In fact, it depends on the application that the simulation has to respect because if for instance an entire tunnel must be studied, it is not necessary to decrease the mesh dimensions to a minimum value of 10 cm while instead in case of a source of smoke, if it is necessary a precise analysis on the distribution of the smoke, the dimensions of the cells must be decreased. Once again, every time that the mesh dimensions are decreased too much, the time necessary in order to perform the complete simulation will increase.

Another problem related to the mesh is also connected with the multiple mesh bodies. FDS requires a sort of alignment when different meshes are present inside of the volume that must be analysed because the junctions must be connected at least every couple of lines. Otherwise in the following situations FDS will provide a warning that there is this misalignment among meshes.

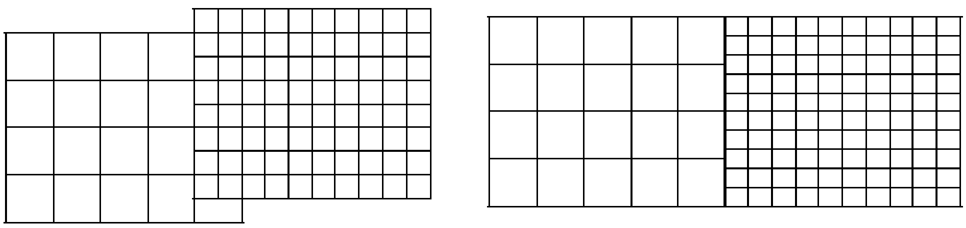

Figure 4.5: Meshes rejected by the software due by misalignments.

Another relevant feature that FDS presents is the fact that it uses explicit time advancement scheme. So for this reason, it is well known that in case of explicit methods, there is a condition of stability that must be respected while for instance in the case of implicit methods this condition is not necessary because they are unconditionally stable. The condition of stability can be obtained representing the solution coming out from the mathematical equation of the explicit method adopted on the Argand-Gauss plane and verifying that nothing is present on the left part of the graph. In the case of the implicit methods, the region of stability is completely empty on the left part of the graph while in the case of the explicit methods what is seen is that there is a restriction in terms of validity. An example could be done with the Euler methods where the solution of stability is obtained as follow:

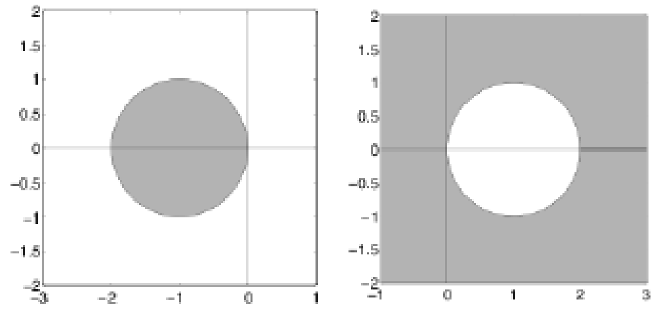

Figure 4.6: Regions of stability for Euler temporal methods.

This is just a simple example of what does it means stability in the case of explicit (on the left) and implicit (on the right) methods. In the case of the temporal strategy implemented in FDS software, the condition that should be considered is the Courant-Friedrichs-Lewy developed in 1928 and it is the method imposed for stability for equation with partial differential derivatives. The main relation is characterised by the following one:

$$CFL = \frac{\Delta t \|u\|}{\Delta} < 1 \tag{4.2}$$

As suggested in Chapter 1, the filtering act performed by the LES simulation considers the cells with an average value of Δ which is simply a sort of balancing phase among the three dimensions of the mesh cells. For these reasons it could be better also to define cubes in meshes when possible because in that case not huge errors are obtained during the filtering phase. So as can be seen by the model, the CFL relation allows to define the time step depending on the speed and on the geometric filter parameter.

The restriction of this is only connected to the upper value but instead there is the possibility to change the lower one diminishing it and yet increasing the time to complete the entire simulation. So, this method allows to tune some parameters in order to decrease the overall time of the simulation reducing for sure the accuracy of the model but without the possibility to find any instabilities. In order to improve the time of the simulation it could be convenient to increase the cell grids and maintain the CFL value closer to 1 as much as possible also increasing the time step after each iteration and decreasing the number of time steps required to end the overall simulation.

4.3.2 Combustion model

As explained in the introduction to the software, one strong point of research on which FDS is focused is connected with the definition of a combustion model. The implementation of the software describes two possible big categories of reactions:

- Combustion: it is connected to the reaction between vapor and oxygen;
- Pyrolysis: generation of fuel vapor at a solid or liquid surface.

From the point of view of the simulation, there are two ways in which it is possible to define the combustion which are the mixing-controlled one and the finite-rate. The second one is not described in this case because it is not relevant for the applications that must be done considering the Frejus tunnel. A simple explanation of that is because FDS by default uses mixing-controlled models because the reactions are infinite and they depend only on the species concentration. In case of finite-rate instead, it is required to apply the simulation with a DNS technique instead of a LES one increasing extremely the time necessary for the simulation.

Describing just a little bit deeper the mixing-controlled model, what is presented is the classical reaction between a fuel and the oxygen present in the air which gives as a result different products and some of them are also really dangerous and this is why it is put this kind of attention on them.

$$C_xH_yO_zN_v + \nu_{O_2}O_2 \rightarrow \nu_{H_2O}H_2O + \nu_{CO}CO + \nu_S soot + \nu_{N_2}N_2 \qquad (4.3)$$

As can be seen, after the combustion process, many elements delivered during the combustion are dangerous for human health. This is the reason why it is very important to have a numerical model able to represent as much as possible the reality of the facts. The additional parameter connected with the smoke production is connected with the amount of fuel particles that are converted in smoke. This is also called smoke fraction and it depends on the type of fuel considered.

4.4 Multiscale approach applied to Frejus tunnel

One of the problems connected with the type of simulations that must be performed on the Frejus tunnel is the time required to get the solution. In fact, depending on the dimensions of the mesh, the time to perform the overall simulation will increase because reasonably, the higher is the number of cells created through the mesh command, the higher will be the number of iterations to solve the NSF set of equations. For this reason, every time that it is required to analyse a complex system as in the case of tunnels, it is preferred to adopt a different technique. Before to explain in reality how the simulation has to be set, it is necessary to define how to solve the problems connected with the time of simulations. The methodology developed in the lasts ten years is also called as multiscale approach and it is used for many tests in case of tunnel simulations because of its efficiency [45, 46, 47, 48]. The main principles that are explained in [48] describes the multiscale approach as a sort of method in which two types of zones are defined when a tunnel is studied: the first one is also called as *near zone* and it is referred to the area in which the fire is located during the simulations while the second is also called *far zone* and here the effects of fire are less sensible and the flux can be considered as fully developed in a one dimensional flow. For this reason what is created is an environment in which only the areas where fire has some effects are analysed in a more precise way while the areas where nothing is changing, they are simply analysed as one-dimensional problems decreasing incredibly the computational costs to get the results. The small portion of the environment that must be studied by the software with the 3D approach is connected with the time of propagation of the smoke. The velocity of propagation of the smoke could be one good result on which base the definition of the 3D space [46].

$$v_{smoke} = c\left(\frac{g\phi(1-\lambda)T}{c_p\rho_0 T_0^2 W}\right)^{1/3} \tag{4.5}$$

Where c is an empirical constant equal to 0.8, Φ is the fire HRR better explained in the following sections, λ is the radiative fraction defined depending on the kind of smoke analysed [49], T is the temperature of the smoke [50] defined considering the admissible limit of persons under firing conditions, W is the width of the tunnel expressed in meters while c_p, ρ_0 and T_0 are defined basing on the environmental conditions.

Once obtained this relation, it is sufficient to multiply the velocity of propagation of the smokes to the critical time in order to obtain where the fire is really effective with respect the overall length of the tunnel. Imposing all the parameters used for the type of simulation it is possible to consider that the overall length of the 3D space and from Eq. (4.1) it is obtained that the mesh size should be composed by cubes with edge dimensions equal to 0,5 m. This will be the mesh and the geometry adopted during all the simulations performed below.

Once defined the 3D space for the computational fluid-dynamic software, it is necessary to focus on the 1D space. About the mono-dimensional approach there are different studies [46, 48]. Here some general points are touched in order to make just a brief recall of what does it means the use of a 1D model in this case. Referring to the equations mentioned in Chapter 1, as can be seen, they are expressed from a generic point of view in the 3D domain. As can be seen for all the three equations (continuity, momentum and energy) many times there is the dependence on the spatial coordinate. In the case of the 1D section, the simplification can be really convenient because instead of considering the partial derivative referred with respect the three main directions, only the one connected with the longitudinal coordinate becomes effective. This reduction in the complexity of the equations decreases for sure also the computational time. For this reason, in general with this kind of methodology it is possible to diminish consistently the amount of time required also to simulate a long tunnel as Frejus is. In fact, it can be proved easily implementing simple simulations that the responsible of the overall time of the simulation itself is mainly the 3D domain. For this reason, with the multiscale approach it is possible to create and study also very complex systems where just one single portion is studied in a precise way but also that small part is affected by the overall 1D network where there could be elements as open doors that communicates with the external environment, fans or only a representation of a part of the tunnel taking into account all the effects connected with that but no fire is present there. This method can be used also for many other sectors such as biological, structural, or other thermal applications for instance [50,51].

The last thing that must be considered when the multiscale approach is adopted is the connection among the 1D branches and the 3D space. In this case two different types of connections can be adopted which are the Dirichlet-Dirichlet boundary conditions where it is necessary an overlapping area in the interaction zone between the 3D space and 1D space. The simulations that will be performed about Frejus tunnel are instead focused on another type of boundary condition which is called the Dirichlet-Neumann one as explained in [46, 52]. In this case, as suggested by [52, 53], in the non-overlapping technique it must be guaranteed the continuity of velocities, pressures and mass flow rate in such a way that the three main laws described in Chapter 1 are always respected. And it is thanks to this principle that the multiscale approach is based because what is done is to use the boundary conditions of the 1D as boundary conditions for the computation of the results in the 3D and vice-versa iteratively until the convergence is not reached and it is possible to move to another time step. The loop of the iterations is well performed in [46], here is briefly explained:

- first the 1D model uses its own boundary conditions to start the resolution of the simple algorithm implemented in the modified version of the software and find the temperature and pressure downstream with respect the 3D space and the temperature and the mass flow rate upstream with respect the 3D environment passing them as averaged values of the sections that communicate with the 1D network;
- solve the CFD equations for the 3D space using the averaged values passed on the previous step by the 1D network in order to solve the CFD problem of the 3D environment;
- Then the averaged values of pressure and the temperature computed in the upstream section of the 3D space can be passed to the connected node while the averaged mass flow rate and temperature computed in the downstream section can be passed to the node at which it is attached.

Continue this iteration until the result obtained for the 3D environment is the same used as boundary condition of the 1D network in the previous iteration. If the convergence is reached it is possible to move to a further step otherwise a new iteration must be performed. In order to get better results on the convergence, a relaxation step is performed avoiding having too sparse results in a very small time instant or very precise results but with a too long time. In addition, it is proved that the boundary conditions between the two spaces are not exchanged continuously but better stability is reached only if the information are passed every 3 seconds between the 1D network and the 3D space. For this reason, the 1D model is solved only every 3 seconds allowing the possibility to the 3D model to reach a certain stability before that its boundary conditions are changed by the 1D network.

It is now possible to move to the definition of how the software works. Starting from the standard distribution of FDS v05, it can be seen that in the software an automatic command is implemented in it. HVAC is the command implemented by the producers of the software in order to make possible the creation of the network [6, 7]. In this case, in order to accomplish to the simulations that must be done on the Frejus tunnel, it is preferred to consider a different method which allows to get better results than the implemented command HVAC in FDS [52].

For the simulations that must be performed it is preferred the use of a software that starts from the basic distribution of FDS but modifying it. The modified software works in collaboration with WhiteSmoke and it allows to make the connection as described before among the 1D and the 3D domain through a new command called as EXCH and EXCH OPEN. In the following section where the case study is analysed in a deeper way, it will be explained how this connection is performed in the precise application of Frejus tunnel.

5. Case study: Frejus tunnel

There are different multiscale approach studies of this technique applied on the longitudinal ventilation systems as literature suggests [48, 52]. Moreover, also some examples about semi-transverse ventilation systems analysis (as the Cuneo tunnel in [52]) and they are just a start for the simulations that will be performed instead in this job because in this case the system in Frejus tunnel adopted is a fully transverse ventilation. The following simulations instead are focused on the analysis and on the description of the ventilation system and how it has to work in case of fire. A common set up for all the tests done is presented in this first part while in the following paragraphs there will be the presentation of the results obtained.

5.1 Simulation of fire in FDS software: HRR

The first thing that must be observed relates to the type of fire considered. In fact, the analysis is done considering a gasoline engine instead of a diesel one because in case of fire it could be much more dangerous with respect a diesel engine machine due by the high flammability of gasoline itself. In the model the gasoline is described considering the definition provided by Carvel. The fire power is simplified considering for each type of transport vehicle what is called a *heat release rate* or simply HRR. This is a value determined experimentally as explained in previous studies made by Carvel [54] on HGV particularly but that are common for many types of vehicles. One thing that must be mentioned before to go further with the definition of the simulations is the fact that during Carvel's tests, it was found a connection between the evolution of the fire and the wind velocity inside of the tunnel itself. In fact, in the research it can be proved that when the speed of the air inside is exactly equal to 3 m/s, the fire growth rate is steeper than other cases in which the speed of wind is bigger or smaller. For this reason, also a differentiation on the evolution of fire could be done depending on the mass flow rate of the natural wind.

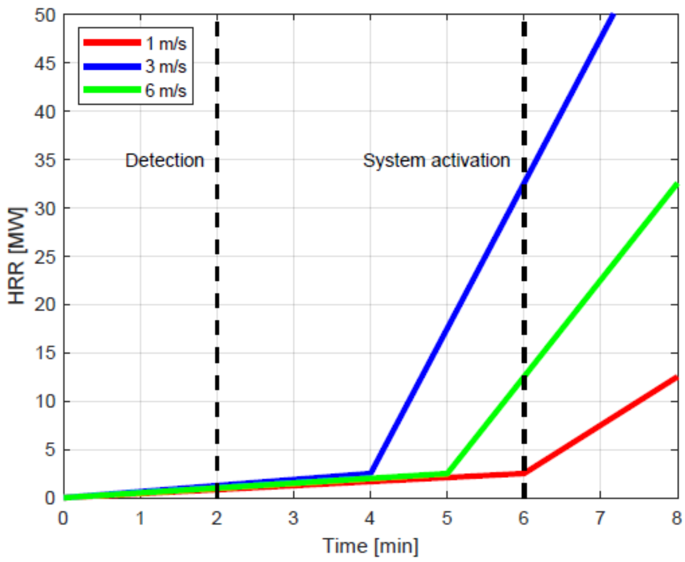

Figure 5.1: Representation of fire growth in an analytical way according to Carvel experimental results.

What is known is that connected with the natural wind present inside of the tunnel, there is the fact that this natural longitudinal ventilation is generated by a pressure difference between the two portals. In general for short tunnels it is not so relevant because if the high of the two entrances is almost the same and not particular changes in the thermodynamic properties of the two environments are present, for sure the pressure difference will be not so effective as in the case of the Frejus tunnel. For this reason, the aim of this job is to analyse what must be performed for the ventilation system in order to act against the effects of pressure confining them in the zone where fire is located. This is exactly the goal: try to explain how it is possible to work in case of fire in a transversal ventilation tunnel acting on the ventilation system and making possible the expulsion of the smokes without the necessity to spread them all along the tunnel.

This is a graphical representation of the three functions adopted to describe the behaviour of fire. There are additional parameters that must be considered when safety plants are designed for every type of system. A delay time is always required from the beginning of the accident and it is composed by two phases which are the detection of fire and finally the reaction of the plant in order to protect the internal environment of the tunnel. For these reasons, as can be seen, these three intervals are described with the two vertical lines and these are the moments in which the smoke start to propagate with a very small reaction coming out from the safety plant. Moreover the experiments made by Carvel show that there is the possibility to find some particular situations in which the growth rate of fire is too much steep and the system has no time to react and to activate the ventilation system before that it reaches the maximum. Otherwise, when the natural wind is high enough, the growth phase is smoother and for sure the maximum of HRR is reached under safer conditions because of the activated fans but the small amount of smoke produced reaches a farer location with respect the fire.

For a car it can be defined that the maximum level of HRR is equal to 30 MW and for this reason, when the simulations reach that high levels, in order to be more conservative and without considering that after some time the fire for sure will decrease because the fuel will end, it is kept constant till the end of the simulation time. The HRR in FDS can be defined as HRRPUA (*heat release rate per unit area)* or HRRPUV (*heat release rate per unit volume)* depending if it is connected to a surface or to the entire volume [7]. In this case it is considered that the upper section of a simulated car will burn and for this reason the HRRPUA is set to 3750 kW. This value is obtained dividing the overall HRR of a car to the surface of a schematic vehicle. In simulations, the fire evolution is imposed through RAMP functions according to Carvel experiments. The car is assumed as a parallelepiped with dimensions of 2x4x1,5 m. Some of the gasoline parameters used in these simulations are taken from [55] and they are useful in order to make more precise simulations. One parameter of particular interest is called SOOT_YIELD and it is connected with rate of production of smoke. In this case it is set to a quantity equal to 0.05.

In general the pressure difference present between the portals makes a natural wind force that travels from the French to the Italian portal due to the fact that the first is at a high of 1228 m from the level of the sea while the Italian one is at a high of 1297 m. What is known is that pressure increases with the decrease of high because the piezometric pressure line must be always the same considering a still gas as it is.

$$Z + \frac{p}{\gamma} = \text{const} \tag{5.1}$$

Knowing the height, it can be computed the level of static pressure referring to an empirical relation as Eq. (5.2).

$$P = 0{,}9877^{\frac{h}{100}} \tag{5.2}$$

Where the unit of measure for h is [m] due by the fact that it is the height of the point in which pressure is computed.
In this case, it is possible that not only piezometric height affects the pressure, but also the environmental conditions such as the temperature or the humidity that can change depending on the seasons. For this reason, what is done is to base the study on the most critical situation obtained from the graph present in Fig. 5.1. The most critical situation corresponds to the faster fire rise because of the necessity of a quick response of the ventilation system. Moreover, as can be seen and also as mentioned before, when the ventilation system reaches the maximum operational conditions, it could be late from the point of view of the evolution of the fire. This is a really good representation of the reality of things because both the sensor limitations and the fan blade inertia when a signal of danger will arrive to the control unit, increase the time necessary to reach the maximum performances of the ventilation system. For this reason, as will be shown below, there is a small instant in which the smoke is not so easy to be controlled.
Another relevant parameter that must be defined at the beginning of the simulations is the density of the air and how consider it. In the simulation performed below, only simple fans are adopted and it can be considered that in case of fans, if the air is the fluid considered, that its density is uncompressible and for this reason knowing the environment conditions it can be computed the density with the equation of perfect gases.

$$\frac{p}{\rho} = R^{*}T \tag{5.3}$$

Where R* is the specific gas constant that in case of air can be considered equal to 286,99 [J/kgK]. Performing this calculation, what is obtained is a density equal to.

This is true only in the sections closed to the fans and this property will be adopted in order to analyse what are the effects of fans when they are activated. In the following paragraph will be explained how this assumption can help during the simulations.

5.2 Network representation for the 1D part of the tunnel

Another aspect that should be touched before to go further is connected with the type of safety system that is implemented in the tunnel is how the far zones are designed. Frejus tunnel ventilation system, as mentioned before, is characterised by three main sections that split it in 3 equal parts almost of 4300 m length. The subdivision in sections is used because in case of fire, instead of spreading the smoke all along the tunnel, it is possible to confine it just in a small portion of the tunnel itself depending where fire is present. For this reason, some control devices must be present for sure inside of the tunnel in order to tune the properties of the ventilation system according to the type of situation considered. These fans are controlled by 4 stations distributed along the tunnel. Two of these stations are disposed on the French side while the other two on the Italian one and they are present in correspondence of the chimneys where also the axial fans are located. This is for practical reasons because the symmetry can help from the ventilation point of view. The subdivision of the network therefore tries to schematise as much as possible the real geometry of the tunnel.

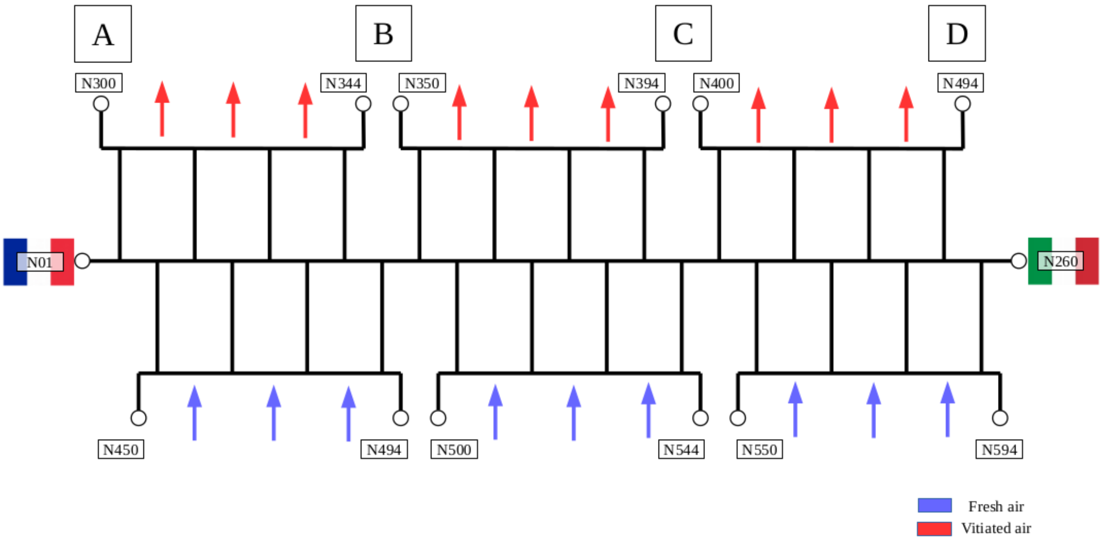

Figure 5.2: Simple scheme of the network adopted in order to simulate the part of the tunnel which is not of interest for the fire analysis.

As can be seen by Fig. 5.2, a really huge number of branches and a really important number of nodes is adopted. In particular, 770 branches are adopted where the indexes to denote them are subdivided as follow:

- from R01 to R260: branches dedicated to simulating the tunnel. For sure, depending on the type of simulation performed, where the 3D domain will be present, the nodes are not considered because in that case it cannot be performed just a one-dimensional analysis there;
- from R300 to R344: branches corresponding to the vitiated air network in between the station A and station B on the French side;
- from R350 to R394: branches corresponding to vitiated air network in between the French station B and the Italian one C;
- from R400 to R444: branches for vitiated air network in between Italians stations C and D;
- from R450 to R494: branches for fresh air supply between French stations A and B;
- from R500 to R544: branches for fresh air supply in between the French and the Italian stations B and C
- from R550 to R594: branches for fresh air supply between Italian stations C

and D;

- from R600 to R750: branches used to connect vitiated air network with the tunnel network;
- from R750 to R900: branches used to connect the fresh air side with tunnel.

This complex network is also characterised by 526 nodes used in order to make possible all the connections among the branches. Different parameters are obtained from literature about the tunnel such as the geometry of all the pipes [55].

	Section [m^2]	Perimeter [m]	f [-]
Tunnel	45	29	0,0027
Fresh air duct	10,8	14	0,013
Vitiated air duct	8,3	12,3	0,014

Table 5.1: Geometrical properties of branches according to tunnel geometry.

Other relevant parameters are connected to the losses. Every time that a fluid moves toward a surface, it feels a certain friction effect, and this should be considered particularly in this case because of the length of the structure. In order to compute the losses that affect the air, an iterative procedure can be performed. In this case the reference method adopted is connected to the Moody diagram where the friction factors are function of Reynold number and of rugosity of the surface.

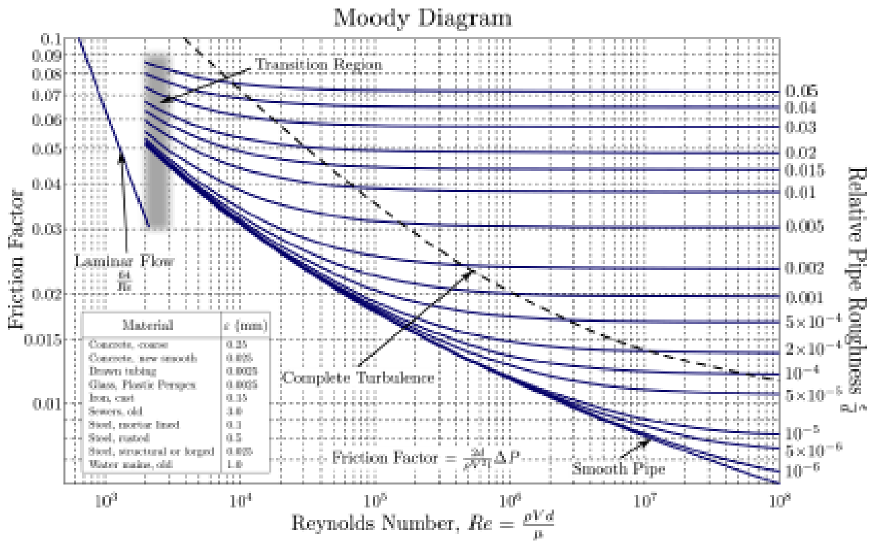

Figure 5.3: Moody diagram for calculation of distributed losses.

The iterative procedure must be performed because it is not known the velocity inside of the tunnel and the friction factor depends on it. For this reason, what can be done is to compute two factors that will be adopted in the following steps which are:

$$D_h = \frac{4A}{p}, \quad \frac{\epsilon}{D_h} \tag{5.4}$$

Then it is possible to enter in the Moody diagram applying the iterative procedure:

- assume a first guessed value of the velocity at the end of the tunnel;
- compute the Reynold number and use this value to find in the graph the amount of friction factor *f*;
- use this factor in order to compute the final velocity at the end of the tunnel;
- verify the convergence:
    - if the velocity is the same of the guessed one, the loop can be stopped;
    - if the two values are quite different, it is necessary to start from the first point.

In the tests performed, the pressure difference considered is characterised by three values that are commonly recorded between the portals and in addition, they are a little bit tuned in order to create a velocity in the three positions of the vehicle tested in order to have almost 3 m/s as velocity which is the fastest growth of the fire as defined by Carvel. During the simulation it is not so easy to have this constant value because of different reasons such as the effects of the sanitary ventilation, the change in pressure that the software computes during the simulation, the change in temperature created by the fire inside of the tunnel, the effects of the dampers that will control the smoke and also due by the fact that fans are simulated through a change in speed at the 1D boundary conditions which means a continuous change of fresh air injected in the tunnel. This last concept is another particularity of the simulations because instead of simulating the fan effects, it is preferred to apply boundary conditions variable in time imposing the amount of mass flow rate that fans have to provide.

5.3 Sanitary ventilation

This first simulation is performed in order to show what happens when no effects are applied connected with the ventilation system. In fact, a homogeneous level of ventilation must be always present in the tunnel because of sanitary reasons. The simulation and the results presented below are simply to show why it is necessary to react to the pressure difference present between the portals. This allows to study better the evolution of the smoke with respect the different situations. The 3D environment is set up to a dimension of 480 m and the mesh size with dimensions of 0,5 m because of the calculations suggested in Eq. (4.1) and Eq. (4.5). The connection between the 1D space and the 3D environment is performed considering the Dirichlet-Neuman previously described. A brief and schematic representation of how the boundary conditions are applied can be shown below.

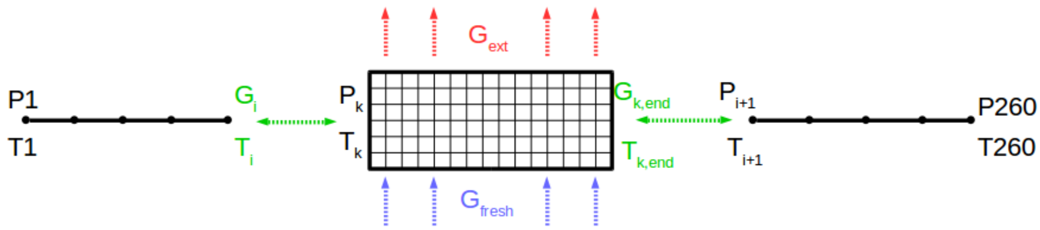

Figure 5.4: Simple scheme of boundary conditions applied to connect 1D and 3D environments.

As can be seen just three pressures are used in this way in order to make the model possible. The most important are the pressures applied in correspondence of the two portals. The entire analysis of the ventilation system is focused on the effects that the ventilation system has to do in order to control the wind velocity inside of the tunnel depending on the pressure difference between the two portals. For this reason, imposing the pressures as boundary conditions of the two extremities is a good way to make good and more realistic simulations. While these are fixed boundary conditions, for the other one, there are different configurations that can be adopted. For the simulations performed in this analysis, due by the fact that many fans must be assumed, what is considered is that wherever a fan should be simulated, a mass flow rate is imposed. This is a quite good approximation of the behaviour of the fans because these parameters are provided by the constructors and depending on the results desired, the fan mass flow rate can be increased or decreased. The interconnection of the part of the tunnel studied with a 3D environment and the simplified one with just a simple 1D network is made through a mass flow rate and a pressure boundary condition on the 1D network. At each time step these interconnections between the 3D and 1D environments are computed continuously and the information between them is performed every 3 seconds.

Once defined the theoretical process behind the software, it is possible to pass directly to the real numbers that should be imposed in the domain. What is considered in case of sanitary ventilation is presented in a precise way in Appendix A. It is obtained through this analytical method the amount of fresh air that the system should be able to guarantee in every kind of situation all along the tunnel. The tunnel ventilation system provides a continuous amount of fresh air that is always present. This is ensured also by the fact that no fan activities are required in order to eject exhaust gases under normal conditions. For this reason, during this first simulation, no effects due by dampers on top of the tunnel are added while for fresh air vents is adopted once again a velocity connection with the 3D environment constant in time. Finally, it can be proved that the amount of fresh air that dampers must continuously provide is equal to the 10% of the overall capability of the fans equal to $1530\ m^3/s$.

The mass flow rate imposed by the dampers instead is always equal to $0\ m^3/s$ when sanitary ventilation is considered because an excess of fresh air is desired inside of the tunnel. The dampers are activated by some sensors distributed along the tunnel and which kind of sensors must be adopted will be explained below. When the fire is detected by the smoke detectors distributed all along the tunnel, the dampers in correspondence of the fire itself are activated. For sure the dampers have always to work with 100% of aspiration because in this way the smoke can be confined strictly where the fire takes place. What should be done, is to make a deeper study on how instead the other two sectors where no fire is present has to react in order to confine better the fire. The maximum mass flow rate that can be ejected by the dampers is equal to $240\, m^3/s$ and they are controlled once again by axial fans. In the simulations performed, the fire is confined in the space of 4 dampers and for this reason during the simulations, on each of them is imposed a mass flow rate equal to $60\, m^3/s$ after the activation.

The activation phase of the dampers is easier than the activation phase of the other two sectors because of informatic limitations imposed by the modified version of the FDS software. Dampers are controlled by smoke detectors distributed along the tunnel roof with distances of 400 m and each of them is connected with the activation phase of the 4 dampers present in the 3D domain. In reality of things, there are also other devices that can be used to control the situation inside of the tunnel such as heat detectors or temperature sensors because the production of smoke is strictly connected with the type of material that is burning. In this case the production of smoke is quite relevant (SOOT_YIELD=0.05) and for this reason it can be proved that the smoke detectors will activate before than sensors focused on the analysis of the temperature. The smoke detectors are in general connected with the visibility concept and if it becomes less than 15 %/m according to PIARC [38], it is necessary to activate the dampers. So, this is the threshold at which smoke detectors are set.

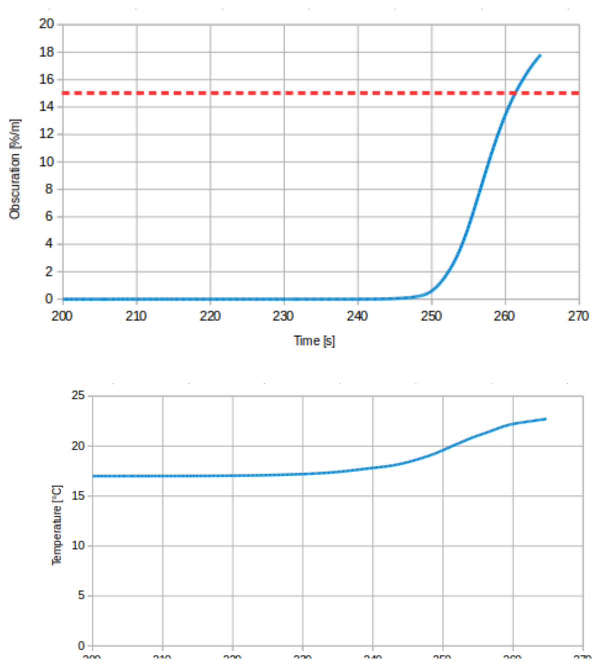

Figure 5.5: Comparison between smoke detectors and temperature sensors recorded data.

This proof that it is sufficient to consider simply smoke detectors as control systems of the dampers. The control system of the sectors where fire is not present are not so easy as said before but for this simple sanitary analysis it can be neglected completely any control system.

Once defined all the boundary conditions useful for the simulation, it is possible to define now the geometry of the 3D environment that will be the same also for the further fire simulations.
Some pictures of how the environment is created in the software are present below and what can be seen is that the tunnel is simplified as prismatic body with a rectangular cross section. This simplification is not too bad because of two reasons:

- in case of Frejus tunnel, a huge part of the section is occupied by the channels on top of it and they limit the available area. Looking at Fig 4.2, this effect is much clear and as can be seen, the prismatic shape is not too far from the reality;
- in FDS it is not possible to make a mesh with strange shapes but only some obstacles with different geometries. For this reason the only way to create the surface is to make an arch characterised by steps but they can be dangerous from fluid-dynamic point of view because they will cause not real concentration effects that modify also the results.

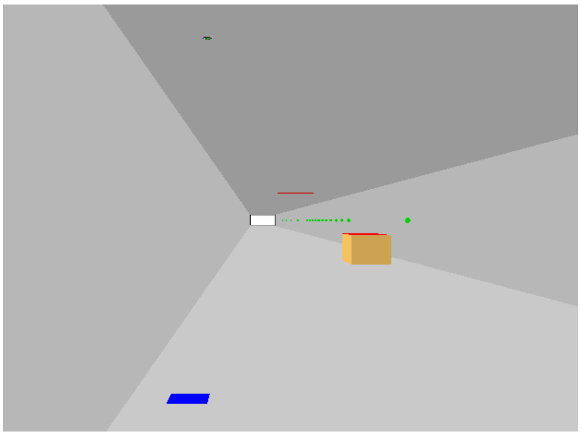

Figure 5.6: Internal view of simulated 3D space.

In Fig 5.6 the blue vents are connected with the fresh air inlets while the red ones represent the dampers on top of the tunnel and on the top of the picture it can be seen also the small representation of the smoke detectors distributed in the 3D environment.
What is obtained by this first simulation is that the smoke is propagating along the tunnel guided by the wind generated by the pressure difference between the portals. In Fig 5.5 a simple wind of almost 1 m/s is imposed through the pressure difference at the portals of 0 Pa but as can be seen, this causes a fast propagation of it because the smoke propagates with a speed of 1 m/s and the sanitary ventilation is ineffective at all to confine it.
Also, the position of the fire is a variable that can affect the different configurations. For the simulation analysed with more accuracy, the vehicle is positioned at a distance from the French portal equal to 6400 m while the pressure difference equal to 240 Pa between the portals causes a wind speed in the vehicle position of almost 3 m/s which is the more critical condition according to Carvel studies.

Depending on the position of the car and depending on the amount of wind imposed, it is possible to get different effects. One of them that is the most critical is also called as backlayering. This phenomenon is the worst from the point of view of longitudinal tunnel ventilation because it means that the momentum generated by the ventilation system is not able to confine the smoke and push it on one particular direction. There are different theories and studies about how compute the speed of wind necessary in order to avoid this effect [46, 52] but in this case the situation is a little bit more complex.

Figure 5.7: Smoke propagation under sanitary ventilation systems with pressure difference equal to 0 Pa.

What is shown in the previous figure, is just an example of what does it means backlayering. In case of Frejus tunnel, backlayering must be analysed differently because the ventilation strategy is completely different with respect common longitudinal tunnels. Depending on the position of the vehicle and where it starts to burn, different ventilation strategies will be adopted because of the fact that the scope is to avoid the propagation of the smoke all around the tunnel and to eject the smoke through dampers put on the top of it. Backlayering is present when the velocity of wind is lower than what is defined critical velocity. The definition of the critical velocity is connected with the smoke propagation velocity. In order to compute the critical velocity, the same relation adopted to evaluate the length of the 3D space can be considered Eq. (4.5). If the velocity of wind is lower than this quantity, it is possible that backlayering will be present as can be seen in Fig. 5.5 where sanitary ventilation conditions are imposed with a velocity of the natural wind equal to 1 m/s. In this case the results obtained for Fig. 5.5 are simply for a demonstration of the effects of backlayering. For this reason, it is not necessary a deeper analysis on it.

The scope of a fully transverse ventilation strategy is to confine the smoke as said before because even if backlayering is avoided, in case of fire in the middle for instance, it means a propagation of the smoke for many chilometers along the tunnel and it could be a problem from many point of view.

For this reason, remembering once again the schematic representation of Frejus tunnel, in Fig. 4.18, different configurations must be applied in the 4 ventilation stations A, B, C and D in order to confine as much as possible the fire inside of the sector where it starts. As can be seen the tunnel is something that can be divided in three main sectors and in brief list below the main groups of operations that should be performed are listed.

- Fire between A and B: being the part connected to the French side, this is the trickiest condition because it is necessary that the other two sectors provide a proper mass flow rate trough the axial fans present out of the firing sector in order to make an overpressure inside of the tunnel that is able to act against the natural ventilation. Moreover, the dampers closed to the fire are opened and the fans connected to them start to eject the smoke from the internal environment to the external one.

- Fire between B and C: also in this case it is not so easy to work because the fire could be exactly in the middle of the tunnel and on both sides it cannot be delivered because in that case both the sides of the tunnel become not accessible at all. What must be done is to consider a sort of cooperation between the upstream and the downstream sectors in order to create a sort of balance between the velocity of the air avoiding the diffusion of fire on both French and Italian portals. What must be done in this case is to evaluate in a very good way the amount of mass flow rate that is provided by fans for fresh air supply in the downstream section being sure that it is not too low with respect the natural wind inside of the tunnel and also it is not too high to push back the smoke on the entrance of the tunnel. For sure in this case the aspiration mass flow rate provided by the dampers present in the middle is important because no other solutions can be used to push out the smoke from the tunnel.
- Fire between C and D: finally, this is the last possible configuration considered. In this case what can be seen is connected with the fact that the closer will be the position of the car to the Italian portal, the safer will be the condition because not only the mechanical ventilation can be used to expel the smoke through the chimneys or through the Italian portals, but also the natural wind generated between the two portals will make safer the situation.

The change in pressure is provided through the simulation phase of fans. In the simulations performed here, fans are considered as the boundary sides of the 1D domain due by the fact that it is not the aim of this job to study and to analyse the behaviour of fans but instead to focus on the smoke evolution in case of transversal ventilation tunnels. For this reason, they are not placed as a part of the 3D environment but just as input parameters knowing the mass flow rate that they are able to provide.

It is now possible to show what are the effects of the sanitary ventilation present in Frejus tunnel. The distribution of fresh air is ensured by the present of some vents distributed all along the tunnel and simulated through the connections of some pipes to the tunnel itself. The mass flow rate is set as constant during all the simulation and different interesting aspects are obtained after this analysis.

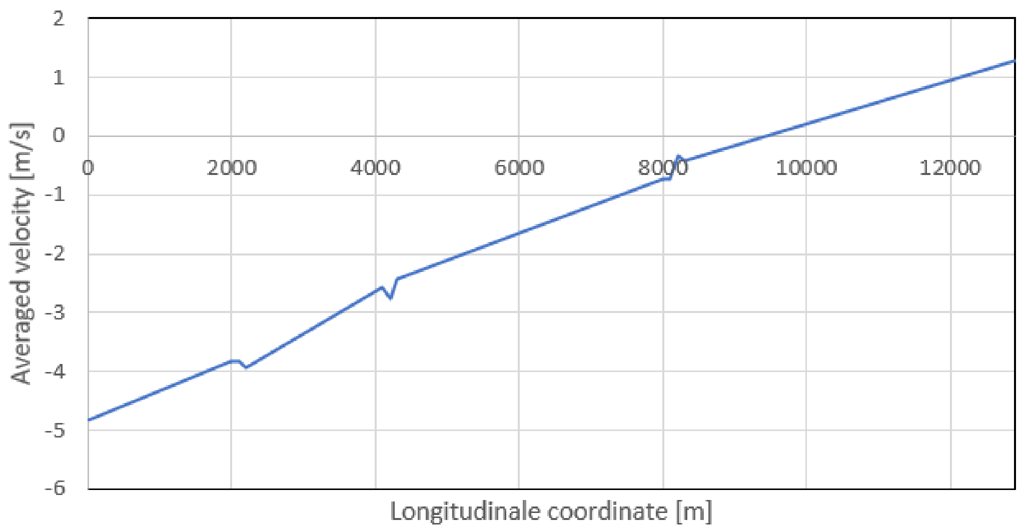

Figure 5.8: Air velocity distribution with 240 Pa of pressure difference between the portals and simple sanitary ventilation.

From the velocity graph, it can be shown that there is a mass flow rate of air that is exiting from both portals. This is done by the fact that the continuous supply of fresh air along the tunnel is not balanced by any extraction phase. For this reason, what happens is that the pressure increases and at a certain point it is higher enough to balance the pressure present on both portals. Direct consequence of this is a section in which for sure the velocity is exactly null but as can be shown by the graph, it is far from the region where the fire is simulated. What is desired is instead to create a section in which the velocity of the air is exactly equal to 0 m/s in correspondence of fire in such a way that the smoke will not propagate but it remains confined in the area of fire in order to be ejected. This is a condition not too easy to be reached because additional effects are created.

In the following tests all the theoretical aspects mentioned till now are demonstrated through numerical results and through 3D pictures which allow to understand better the complexity of the problem. What will be done during the simulations below is to run many cases for each position of the car inside of the tunnel in order to see what happens to the flow of air. The aim of the following steps is to analyse what are the conditions that must be imposed on all of the four ventilation stations in order to reach a condition in which the smoke is produced and confined directly closed to the car without any spreading condition of it. The closer is the smoke to the car, the better will be the configuration so it means that if the velocity of the air in correspondence of the vehicle is equal to zero, the configuration can be considered as safe. In addition, graphs related to velocity profile and temperature distribution during the simulations are presented.

5.4 Ventilation system analysis

Taking into account the case study described in the previous point, as shown, in case of sanitary ventilation the smoke is not confined by the four dampers as necessary to make the tunnel safe. The next step that can be considered is connected with the activation of dampers. The activation phase of the system requires some time and this is connected with the smoke detectors sensibility. Considering the threshold of sensibility of the smoke detectors, the dampers are activated after a time equal to 300 s when the pressure difference is equal to 240 Pa.

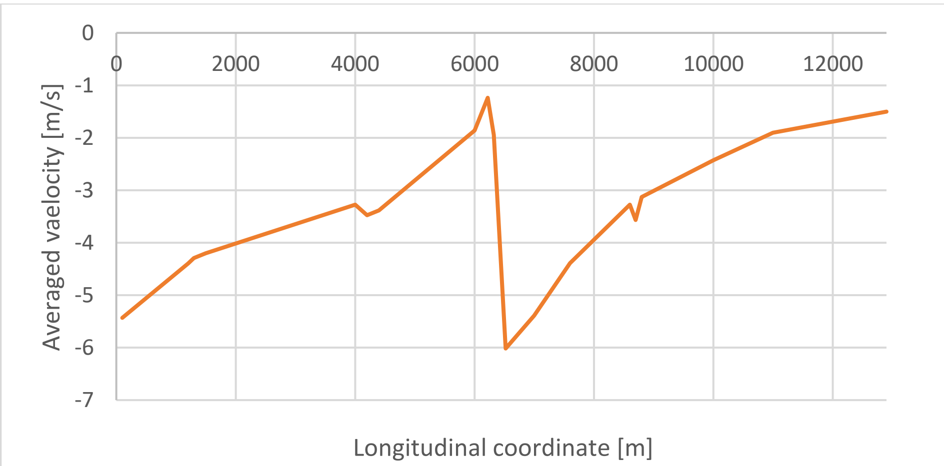

Figure 5.9: Effects on the velocities with 100% of extraction in the fire zone and 240 Pa of pressure difference between the portals.

The effects of the activation of the dampers cause a huge mass flow rate exit from the section in which the fire is situated and this means a change in the velocities. As can be seen, due by the relevant pressure difference among the portals, the aspirated mass flow rate of air is not sufficient to make a zero-velocity section as desired and the smoke will propagate to the French portal. For this reason, a new solution must be adopted in order to confine the smoke also when the simple extraction system is not able to do anything.

The solution to the previous problem is adopted also for other kind of configurations. Another interesting case that can be analysed is the case in which the pressure difference between the portals is null. In fact, till now a pressure difference between the portals equal to 240 Pa was considered. If it is considered as equal to 0 Pa, what is obtained is a different situation. Here the mass flow rate of air is simply done by the sanitary ventilation present inside of the tunnel and by the smoke produced by the fire and the extraction provided by the dampers is sufficient to get a zero-velocity section that must be controlled in position. The graph of the velocities obtained when dampers are activated without any pressure difference shows a zero-velocity section as in Fig 5.10.

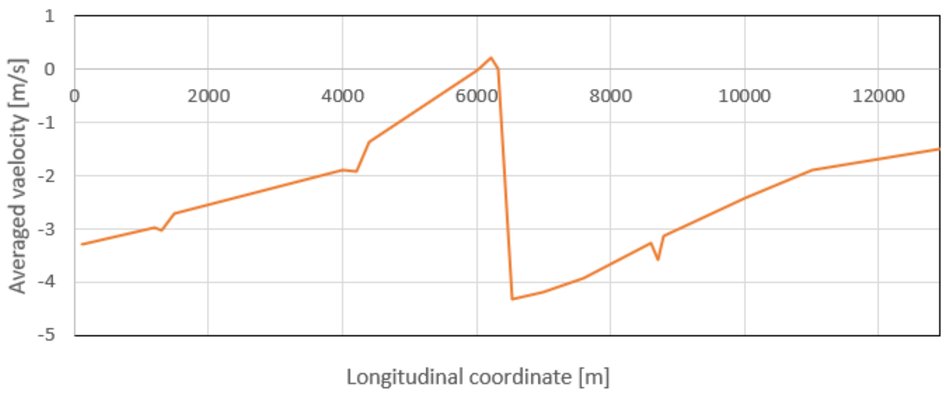

Figure 5.10: Effects on the velocities of 100% of extraction in the fire zone and 0 Pa of pressure difference between the portals.

The main problem connected with this kind of configuration is linked with the previous propagation of the smoke. As can be seen by the graph of the velocities, the zero-velocity section is shown in correspondence of x=6200 m but the fire location is at x=6400 m. During the analysis of the results of this first preliminary step it was obtained some critical results. First is connected with the location of the zero-velocity section. When this condition is not located exactly where the fire takes place, it is not possible to confine the smokes produced in a proper way but they will propagate according to the velocity field. Moreover, a limitation about the reality of the safety plant must be considered. In fact, from the starting phase of a fire to the time in which the ventilation system is really effective, many delay phases must be considered connected with the detection of the fire by the sensors and with the inertia of the devices and of the air. For this reason if the smoke is able to overcome the zero velocity section before of the effective aspiration level of the plant, part of the smoke will be propagated along the tunnel while another part remains in the area in which the velocity is null without any possibility to be eliminated. These two conditions make the simple aspiration solution not sufficient to control the smokes.

These two cases analysed before are just a simple representation of the complexity of the limitations that a simple extraction of hot gases is not the proper thing that must be performed.

For these reasons, what the ventilation system must be able to do is to confine the smokes also in these critical cases. In order to do this, the sectors far from the fire are adopted providing the correct amount of fresh air in order to create an overpressure in correspondence of the section after the zero velocity one that pushes back the smoke closed to the fire position and make possible the ejection through the dampers in correspondence of the fire itself.

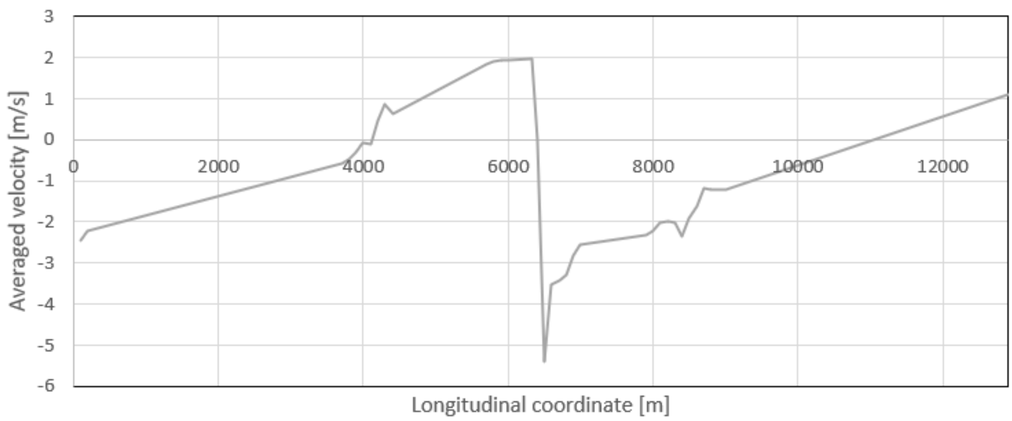

Figure 5.11: Final velocity with supply fans activation with 240 Pa of pressure difference between the portals.

What is obtained inside of the tunnel is that the smoke propagates for a certain time and when the fire is detected by smoke sensors, both dampers and supply fans are activated in order to confine it.

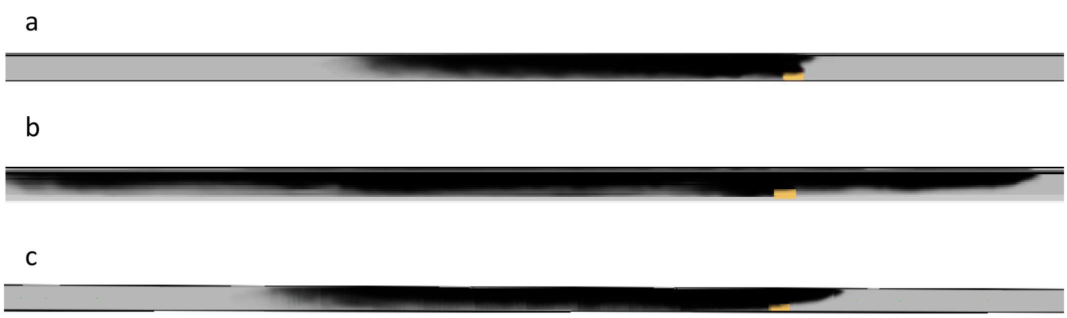

Figure 5.12: Smoke propagation with a pressure difference between portals equal to 240 Pa.

The three pictures represent:

a) 120 s: detection phase and activation of dampers;

b) 300 s: slower propagation caused by extraction of dampers;

c) 400 s: complete confinement of smoke with fresh air supply of the other two sectors.

In Fig. 5.12 it is possible to see the evolution of the smoke according to the ventilation system tuning phase. The configuration is exactly the same as before where the pressure difference between the portals is 240 Pa and the position of the car is always equal to x=6400 m. The smoke is able to propagate passing over the dampers that must be adopted to confine it. After the detection phase (after 120 s) of the smoke detector considered, the four dampers connected to it are opened creating a sudden decrease in velocity field inside of the 3D environment. What can be seen is that after the simple activation of the dampers, a zero-velocity section downstream with respect the 3D environment is always present. This means a very critical situation because the smoke that overcomes the 3D environment will not be confined and aspirated by the dampers activated and it will travel on the x=0 m direction. In order to be sure that the smoke is pushed back to the correct dampers, after some simulations, making some tests, it is obtained how to work with the external sectors with respect the part of tunnel analysed in FDS. In the case described till now, after some proofs it was obtained that the sectors closed to the French portal provide an additional amount of fresh air equal to 30% of the overall capability of the fans. It is necessary to act also against the backlayering effect that happens after the activation of dampers because when dampers are working at 100% of them power, the velocity of wind in between them will decrease due by the high extraction level. This decrease in the speed of the wind causes a relevant backlayering effect because the velocity decreases below the critical speed defined as threshold value.

In order to understand better the effects of the ventilation system, it is possible to plot the velocity just focusing on the 3D environment. In the following graph the velocities of the three sections are plotted and what can be seen is that with only sanitary ventilation, a little increase in the velocity is imposed because the continuous supply of fresh air without any extraction, it causes an increase in the velocity when French portal direction is considered. The second line instead is connected with the localized extraction applied by the dampers. In the case of dampers, a general delay time of 240 s can be considered if no control device is present. For the simulations considered, it is preferred to use these kind of control systems because as presented bin Fig. 5.12 are more sensible to the presence of smoke. The smoke detectors are equally distributed along the tunnel: almost 1 each 500 m [31]. In the case analysed the smoke detectors are activated after 263 s almost and after 40 s it is obtained a stable configuration where once again the zero-velocity section is not obtained due by the high-pressure difference between the portals. In the end what is done is to tune the velocities in correspondence of the region in which the fire takes place in such a way that the smoke produced is not propagated along the tunnel but it is confined in a close environment instead.

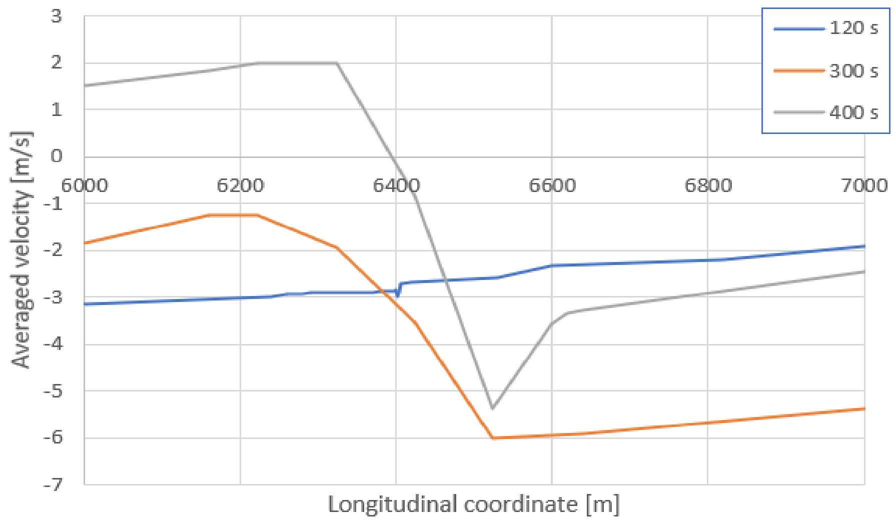

Figure 5.13: Focus on the velocity behaviour depending on dampers and fans conditions with 240 Pa of pressure difference.

As can be seen, some particularities of the graph are described here. The line corresponding to the time step 120 s is something similar to the commons straight decreasing line described in Fig. 5.8 but in this case the fire affects its behaviour increasing a little bit the velocity downstream of the fire location. This small drop in speed is in fact connected with the fire presence there. If the same configuration is tested without any fire, the line plotted will be constant in slope. The change of the slope is not so relevant because remembering once again HRR behaviour, it is possible to see that when the detection takes place, the fire has not a relevant power. For the remaining two lines instead, the orange one describes the behaviour of the velocities just before supply fans activation. As can be seen, when the dampers are working with 100%, it is not sufficient the amount of air extracted in order to obtain a zero-velocity section in that precise point. After 400 s also the additional supply of fresh air becomes effective and what can be seen is that the downstream sectors (FA2, FA3 and FA4) that for the case study are working with 35% of additional supply of fresh air and fans upstream with respect the fire (FA4 and FA5) that instead have to work with 100% of supply of air because otherwise a the smoke overcome the damper zone and the zero-velocity section is presented outside with respect the fire location creating problems related to aspiration of the smoke.

The last interesting result that is defined as another control parameter in order to make safe the tunnel environment in case of tunnel, according to PIARC standards, is the temperature. For this reason, considering the three relevant time steps as for the velocity profiles, the following picture shows a controlled temperature distribution even if it is a little bit spread around 300 s.

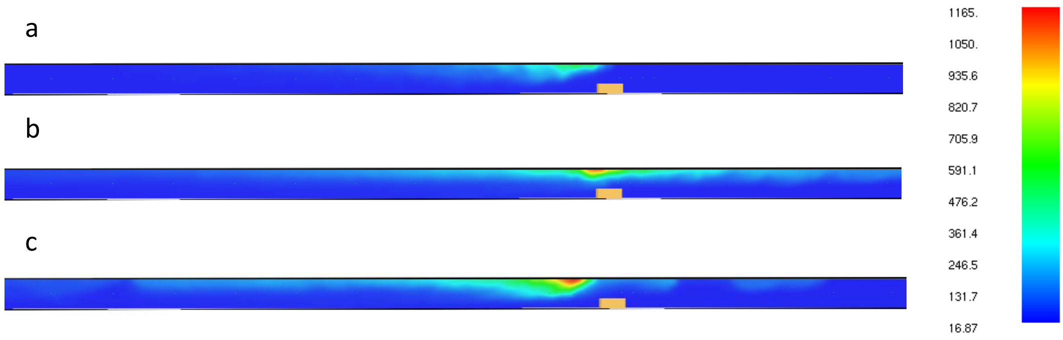

Figure 5.14: Temperature evolution with pressure difference equal to 240 Pa and at the time steps of 120 s, 300 s, and 400 s.

Once described how it is necessary to act in order to confine the smoke, different kind of situations can be studied trying to impose different levels of fresh air from the sectors not interested by fire. After different simulations only the relevant results are presented below considering as variables the pressure difference between the portals and the longitudinal position of the car. A clearer scheme of the ventilation system is provided below in order to express easier how the ventilation system has to be set depending on the configuration that could be present.

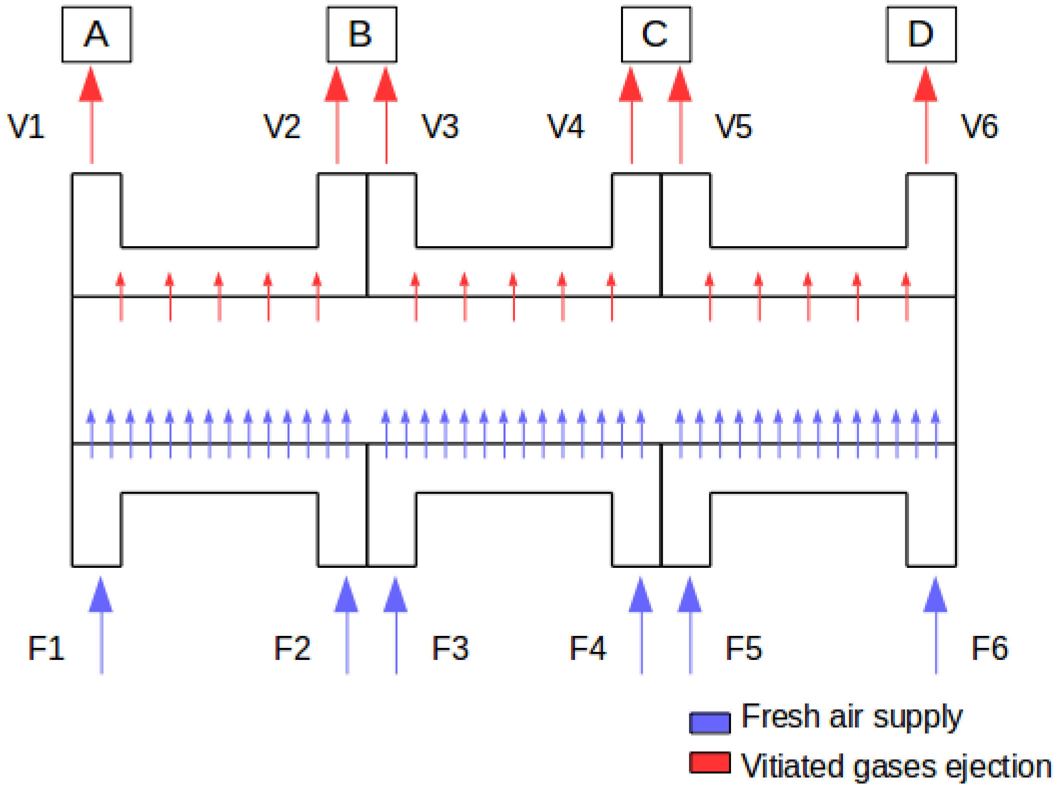

Figure 5.15: Description of a schematic representation of the ventilation stations in simulations.

Many of these ventilation strategies can be obtained by [55] where the results are obtained performing a fuzzy logic procedure while other of them are proved after the simulations performed during these analyses.

x_F	Δp [Pa]	V1	V2	V3	V4	V5	V6	F1	F2	F3	F4	F5	F6
2150	0	100	100	100	100	0	0	70	0	0	0	0	0
2150	240	100	100	100	100	0	0	80	0	0	0	0	0
2150	-200	100	100	100	100	100	0	100	0	0	0	0	0
6400	0	0	0	100	100	0	0	30	30	30	0	0	0
6400	240	0	0	100	100	100	100	80	80	30	30	30	30
6400	-200	0	0	100	100	100	100	80	100	30	30	30	0
10700	0	0	0	0	0	100	100	0	0	80	80	0	0
10700	240	0	0	100	100	100	100	0	0	50	50	0	0
10700	-200	0	0	0	0	100	100	0	0	40	40	0	0

Table 5.2: Summation of all the test case analysed with the percentage of activation of each fan in order to confine the smoke.

Taking into account once again the case considered before (position of the vehicle 6400 m from French portal and pressure difference equal to 240 Pa), other kind of variables can be shown in order to explain better what are the results adopting the configurations described in Table 5.2.

What is obtained after a huge number huge number of simulation is that not only the sectors where the fire is taking place are responsible of the controlling phase of the smoke but the entire ventilation system of the tunnel must be able to work together in order to confine smoke. The table above considers only three particular position of the vehicles because in this case a location in correspondence of each of the three sectors is considered. Moreover, only three significant levels of pressure are considered. The most critical in terms of smoke propagation is for sure the case in which the pressure difference is equal to 240 s because a steeper increase in the fire evolution is presented before with respect the other situations.

6. Conclusions and future works

After all the analysis performed for the previous considerations, what can be seen is that the ventilation strategy is not so easy to be analysed when long tunnels like Frejus are considered. It is necessary to focus the analysis on some particular case study trying to describe all the possible configurations that can be seen. Many times, it is sufficient to focus simply on some generic cases to define in a precise way the behaviour of the system but many other times, this is not possible. The effects that can change the variables inside of the tunnel are many such as the fire power, the pressure difference, the vehicle position, the traffic conditions. All of them during the analysis performed are collected under some particular cases but this is not always possible.

The modified version of the software guarantees to obtain in reasonable times the results of the simulations studying with the precise level of accuracy only the areas that require this high level of precision. But the main advantage is that this methodology allows to obtain also results about the overall behaviour of the structure and the interaction between the near and the far zones make possible also the consideration of some relevant devices that cannot be neglected such as fans. The Frejus tunnel is one of the most complex tunnels in terms of ventilation strategies and also in terms of complexity of the network because once again the excessive length and the high difference in the environmental conditions present at the two portals can create very critical effects. Even if these critical conditions are present, the software is shown as solid, fast and also accurate because according the results obtained by the simulations are what was expected. About the additional simulations that are added with respect the older results, the behaviour of the fluids inside of the tunnel is always in accordance to what is expected.

About the control phase of the ventilation system, The Frejus tunnel allows to automatically activate the dampers and the supply fans according to sensor detections and to the thermodynamic variables. In addition the safety plant guarantee the possibility to be controlled also manually because once again there is the possibility that some critical accidents happen inside of the tunnel such as explosion conditions or other dangerous events and for this reason, it is not sufficient to focus simply on the small number of cases considered. During the simulations, as mentioned before, it is not possible to create a really controller able to check all the properties during the simulation modifying the ventilation system in order to confine them because of the software limitations. This could be one weakness of the software because as explained before, the boundary conditions applied to the 1D environment, can be simulated simply as RAMPs so variable in time and no other control commands can be passed to this command. For this reason one possible way to improve the accuracy of the simulations and also to give precise data about the system, could be to modify the input file of the one-dimensional network in such a way that also fans located at the boundaries of it are able to react according to what is happening inside of the 3D environment where fire is present. For this reason, no transient behaviour of supply fans can be analysed because of this software limitation. One way to improve it could be the possibility to apply a PID controller in order to create a retroactive system able to make in communication the 1D and the 3D environment not only at the interface of them, but the overall network can communicate and react according to the proper strategy.

Another relevant aspect that could change the real results with respect the simulated ones is connected with the shape of the geometry. This could create some weak points and some relative errors because of the difference between the real geometry of the Frejus tunnel and the simulated one. In the last years a new software able to create precise geometries is developed and it is called as Pyrosim. The software is able to pass from a CAD file created by the designer directly to the input file version useful in FDS. For the simulations performed it is preferred to use a simplified geometry because what must be analysed is the overall behaviour of the entire system. As a consequence in the end it can be said that the software developed allows to obtain reliable results but the research focused on this field cannot be stopped here because there are still some aspects that must be corrected.

Appendix A - Design of ventilation systems

The first thing that must be done in order to design a tunnel is the definition of the ventilation system under the normal working conditions. One of the most important aspect that must be considered in this case is connected with the traffic conditions because in the case of a congested traffic, for sure, the level of emissions will be different with respect the normal traffic operations.

A1. Dangerous pollutants for human body

Three main dangerous agents must be considered when ventilation system is designed which are CO (carbon monoxide), particulate matter and NO_x that embeds both NO and NO_2. Those substances are really dangerous for different reasons and for sure an appropriate system of ventilation is used to spread them in the external environment without creating an accumulation inside of the tunnel.

The carbon monoxide CO is really dangerous for the human health because it is imperceptible due to the absence of colour and smell but when it is breathed with a concentration higher than 5 mg/m^3, it becomes a problem as suggested by the ministry of health. The main problem is that the carbon monoxide is able to connect itself with haemoglobin which is the responsible of the oxygen transport in the human body and this connection is 200/300 times stronger than the connection between oxygen and haemoglobin making the protein ineffective for the oxygen transport. This dangerous gas is created by an incomplete combustion due by the absence of an adequate level of oxygen of it as in the case of the presence of vehicle. In the case of nitrogen oxides, it is not so easy to distinguish between the nitrogen oxides NO and nitrogen dioxides NO_2 and as a consequence usually it is defined the amount of generic NO_x and, after some experimental tests, know the exact percentage of NO_2 present inside of it. Today some sensors were invented to define the amount of NO_2 but for difference reasons they are not easily used in tunnels. The first reason is due to high costs of the sensors because they are electrochemical cells or sensors based on optical measurements; the second reason is connected with the maintenance requirements for these sensors and finally the third reason is connected with the accuracy of these sensors because they are able to measure the level of NO_2 with a precision of 0.1 ppm but sometimes it could be required to be more precise than this. In the lasts years, mainly in France, some of these optical sensors were installed in different tunnels and they show good results but it is too early to get conclusions from them because it is better to see the effects in the long run. So as said before, could be better the evaluation of them through experimental tests made in controlled environments able to measure in a precise way the amount of NO_2 in the air and after that make some tables in which it is reported the percentage of it depending on the type of vehicle.

NO_2/NO_x ratio	PC gasoline	PC diesel	LCV gasoline	LCV diesel	HGV
2018 (base year)	0.05	0.33	0.05	0.32	0.11
2030	0.05	0.31	0.05	0.31	0.21

Table A.1: Level of NO2 depending on different type of vehicles.

Another important factor is that the standard is referred with respect a base year which is selected as the 2018. What is relevant is that the European standard are much more restrictive with respect the past years and for this reason, for sure, the years following the base year will have production of all of the pollutants that will be always less than the previous ones. As said before, the pollutant considered during the design of a tunnel is mainly connected with the NO_2 because NO is not harmful at the encountered levels while the other is because it can irritate the lungs and lower the resistance to respiratory infections. Moreover, it creates some dangerous gases such as ozone, nitric acid and nitrous acid.

The last dangerous pollutant relates to the particulate matter (PM) emissions and problems related to the visibility. When PM is considered, two types of particulates must be considered which are the exhaust and the non-exhaust ones. In the first case, it is referred to the emissions coming out from the tailpipe of the vehicles so once again related directly to the emissions while in the second case it refers to the particles coming out from the tyre consumption, from the road and all the other materials which deteriorates. All these particles can create effects of reduction of visibility because the light, every time that encounter a particle, suffers three phenomena which are absorption, transmission and reflection. For sure when the particles are less, these effects are not visible almost at all but when we are referring to a high level of PM, the visibility becomes a problem. Many times, in tunnel design the restrictions imposed by the PM are not a problem as will be proved in the following steps.

A2. Steps for the evaluation of the emissions

The first step considered is the evaluation of the traffic conditions. These data are gotten by the Sitaf Spa which stands for *"Società Italiana per il Traforo Autostradale del Frejus"* and every month monitories the average traffic per day that flows inside of the tunnel.

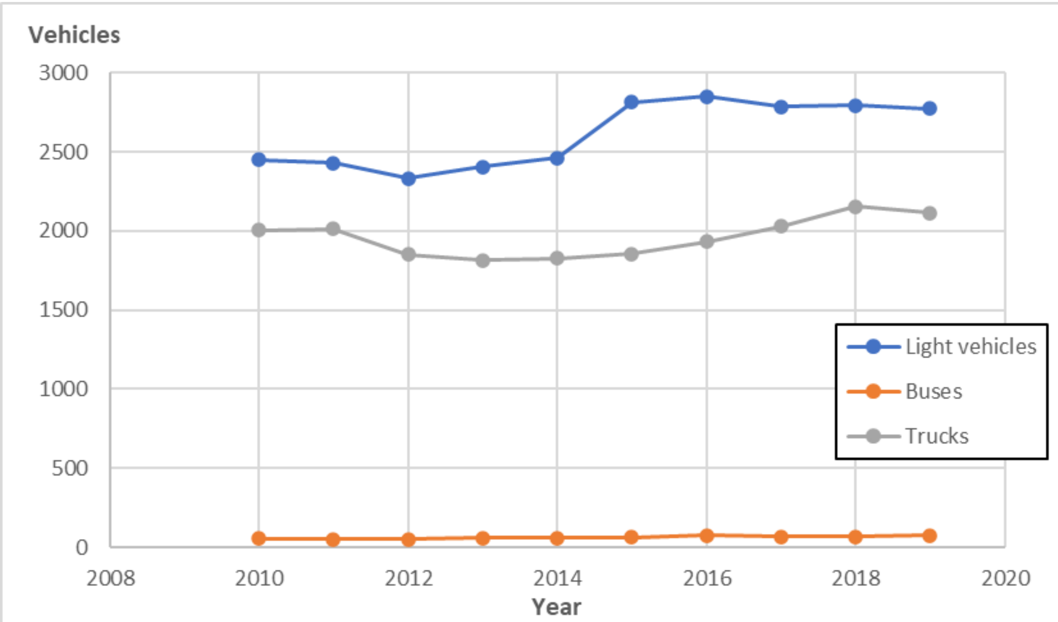

Figure A. 1: Traffic evolution during the years through Frejus tunnel.

As can be shown, there are not relevant changing during the years related to the fleet of vehicles. In order to design the ventilation system, it is required to consider two most important formulas which are used to describe the amount of pollutant produced as an average value in normal traffic conditions. The design condition for normal traffic flow is considered for the flow of vehicles that is more critical. In this case, it is connected with the 2018 again because as suggested also by the table below, it is the year in which vehicles travelling inside of the tunnel were much more present.

Daily average traffic during the years			
Year	Cars	Buses	Trucks
2010	2449	53	2004
2011	2430	51	2013
2012	2331	52	1852
2013	2406	59	1816
2014	2461	59	1826
2015	2812	64	1855
2016	2848	73	1930
2017	2783	67	2029
2018	2795	67	2154
2019	2771	73	2114

Table A.2: Data provided by Sitaf of traffic conditions in Frejus tunnel during previous decade.

Once defined the traffic conditions used to compute the amount of emissions, it is now possible to move to the real calculations that will be used for the computation of the production of emissions and as a consequence used for the computation of the amount of fresh air under common working operations of the ventilation system. The two main relations that must be used are connected together.

$$G_{sec} = \sum_{i=1}^{N_{cat}} g_{fac,i} n_{fac,i} \tag{A.1}$$

$$G_{tun} = \sum_{k=1}^{N_{lane}} \sum_{j=1}^{N_{sec}} G_{sec,j} \tag{A.2}$$

The first relation is a simplification of the calculation which is based on the analysis of one simple part of the tunnel instead of evaluating all the entire system and after that it is possible to use that unit in order to compute the total emissions for the computation of the flow of fresh air.

The vehicles must be distinguished between two main categories which are the passenger cars (PC) and the heavy-good vehicles (HGV). As suggested also by the standard it is better to consider both trucks and buses in this category. Moreover, an ulterior distinction between the diesel and gasoline cars must be done because for sure the type of emissions and also of pollutants will be different. For the overall calculation, vehicles of class A are considered.

For the first computation of the emissions it is possible to consider normal traffic conditions. For this reason, a speed of 60 km/h is selected for the designing phase of the ventilation system. as suggested by the previous relations, the section on which the traffic is analysed is equal to 1 km of length with just one line in such a way that it is possible to repeat the results obtained from this small section. Considering the data presented in the previous table, the 2018 is selected not only as the base year, but also because the daily average flow of vehicles during that year was the largest of the last ten years. The ventilation system in the end is designed considering that ventilation flow. The first parameter that can be computed is the flow of cars for each hour simply evaluated with the following relation:

$$q = \frac{n_{vehicles}}{24h} \tag{A.3}$$

And from this, the density of vehicles for each km is obtained as:

$$D = \frac{q}{v_{average}} \tag{A.4}$$

From these calculations the density of vehicles for each km is equal to 5.026 veh/km. The fraction of HGV and PC with respect the total can be easily obtained just dividing the two categories by the total amount of vehicles which travel through the tunnel each day. In this way applying the following relation it is possible to get the number of vehicles for each of the categories that are present in each km in the tunnel.

$$n_{HGV} = L_{sec} a_{HGV} D \tag{A.5}$$

$$n_{PC} = L_{sec} a_{PC} D \tag{A.6}$$

Vehicle in the tunnel section			Total number of vehicles		
n_{HGV}	$n_{gasoline}$	n_{diesel}	n_{HGV}	$n_{gasoline}$	n_{diesel}
2,225	1,400	1,400	28,635	18,018	18,018

Table A. 3: Total number of vehicles present in the tunnel basing on statistical analysis.

Those numbers have no meaning from the physical point of view because they represent vehicles with a comma value but for the results that must be achieved, they can be considered as they are. On the right hand of the table, it is possible to see the overall number of vehicles that cotemporally are travelling through the tunnel and this gives the possibility to compute the amount of pollutants produced by these vehicles. For sake of simplicity, the PC are subdivided equally between gasoline and diesel engines because it is possible to say that they are equally spit among these two categories.

All the calculations obtained from these numbers are connected with the fact that the Sitaf provide precise data about the amount of vehicles that are travelling through the Frejus tunnel because in this case it is not the case in which there is the design of a complete new tunnel but it is just an application on an existing one on which some firing conditions want to be studied. For this reason, the approximated values of the flow of vehicles presented in the table in the PIARC standard are not efficient and not used.

In order to compute the emissions of all these vehicles, it is possible to use a new relation presented always by the PIARC method which describes the possibility to define the pollutants using a mathematical relationship that embeds all the factors of interest.

$$g_{fac} = g_{cat} f_t f_m f_a + g_{non-ex} \quad \text{(A.7)}$$

Each of this factor can be explained:

- g_{cat} is a factor that describes the emissions of each category of the vehicles (in this case only PC with gasoline and diesel engines and HGV) depending on two parameters that are the slope of the tunnel and the average speed of the vehicle. The average speed considered in this case is 60 km/h while the slope is given by the drawn related to the tunnel which is equal to 0,54%;

- $f_t$is called also time factor and it considers the fact that more stringent emissions standards are present in the last years and for this reason if the year considered is bigger than the base one (2018), this factor will be lower than 1 because the emissions decrease in time;
- $f_m$is the weight factor and it describes the dependence of the emissions on the weight of the trucks. Once again there is a standard weight under which there are no effects on it changing. This value is experimentally set at 23 ton. In the following calculations it is considered an averaged value of 23 tons in order to balance the absence of LCV (light commercial vehicles);
- $f_a$is the altitude factor. It is possible to prove that vehicles are affected by the factor of altitude and some experimental tables can be used in order to do this but it is not so easy to define in a precise way this and as a consequence it is better to consider in a conservative way these factors;
- $g_{non-ex}$is a factor that must use only connected with the visibility problem due by the fact that it has no effects if connected with gases produced by the vehicles.

All the tables used in order to compute these factors are present only for particular values of inclination and speed. Particularly, referring to the inclination of the tunnel, it is required a linear interpolation because factors are not listed for an inclination of 0,54% that is the specific slope of Frejus tunnel. Moreover, it must be considered that the factors are listed for the three most important pollutants produced by the passage of vehicles which are the CO, NO_x and PM as explained in the previous paragraph. After linear interpolations, the data obtained are presented below.

CO	g_{base} [g/h]	f_h	f_t	f_m	g_{fac} [g/h]
PC gasoline	20,117	1,1368	0,91	1	20,8107951
PC diesel	1,816	1	0,92	1	1,67072
HGV	39,868	1	0,89	1	35,48252

NOx	g_{base} [g/h]	f_h	f_t	f_m	g_{fac} [g/h]
PC gasoline	4,005	1	0,85	1	3,40425
PC diesel	26,951	1	0,87	1	23,44737
HGV	138,911	1	0,71	1	98,62681

Opacity	g_{ex} [m^2/h]	g_{non-ex} [m^2/h]	f_h	f_t	f_m	g_{fac} [m^2/h]
PC gasoline	0,327	6,7	1	0,98	1	7,02046
PC diesel	3,07	6,7	1	0,76	1	9,0332
HGV	10,65	30,3	1	0,96	1	40,524

Table A. 4: Coefficients used in order to compute the level of emissions and of reduction in visibility for different categories of vehicles.

In the case of the emissions of noxious gases, the g_{fac} is computed just multiplying all the terms in each row while in the case of problems related to visibility, it is required not only to multiply all the factors for the g_{ex} but also to add the contribution to the reduction of visibility provided by the g_{non-ex} gases. In the end, in order to define the flow rate of dangerous gases, it is simply necessary multiply each g_{fac} per each vehicle connected. In this case the relation presented at the beginning is a little bit changed because it is not considered the density of the vehicles for each line but directly the overall density through the tunnel and for this reason it is not necessary to multiply also by the number of lines even if it is considered a bidirectional tunnel. In the end the amount of gases computed are described in the following table.

G_{tun}	CO [g/h]	NO_x [g/h]	Opacity [m^2/h]
PC gasoline	374,96	61,34	126,49
PC diesel	30,10	422,47	162,76
HGV	1016,04	2824,19	1160,41
Total	1421,11	3307,99	1449,66

Table A.5: Flow of dangerous gases and production of PM in Frejus tunnel.

A3. Amount of fresh air required by the ventilation system

Finally, it is possible to compute what is necessary for the design of a tunnel, the amount of fresh air required to eliminate the pollutants in the tunnel during common flow of vehicles. The main relation that is used is the following:

$$Q = \frac{G_{tun}}{C_{adm} - C_{amb}} \tag{A.8}$$

In this case two values must be defined which are the admissible limit and the environment conditions. For the second one usually the Piarc convention suggests adopting values equal to 0 because even if they are not, the noxious particles are spread in the environment and they are really small in number and for this reason they can be neglected. What is relevant instead is the admissible limit of each of these elements. Once again there are some experimental values obtained following the PIARC standard because they depend on the human body conditions.

Traffic situation	Design value		Operation condition	Operation limits
Free flowing peak traffic 50 – 100 km/h	70 ppm		Normal operation*	< 100 ppm
Daily congested traffic, stopped on all lanes	70 ppm		Planned maintenance work in a tunnel under traffic**	20 ppm
Exceptional congested traffic, stopped on all lanes	90 ppm		Threshold value for tunnel closure	200 ppm

Table A.6: Limit value for the level of CO inside of the tunnel.

In this case once again a simple flowing of the traffic is considered and the limit as a consequence is imposed to 70 ppm of CO inside of the tunnel. In the case instead of the amount of NO_x admitted, it is required a higher level of it equal to 1 ppm almost because of toxicity and strong problems connected with human health. In some other countries the level of it is decreased to 0.5 ppm. Once obtained these two limits, it is required to express them in [g/m^3] in order to be consistent with the mass flow rate of the gases in the tunnel. In order to do this, it is necessary to multiply the admissible limit expressed in ppm by the density of the gases expressed in kg/m^3 and convert it in the unit of measure desired. In this particular case:

$$\rho_{CO} = 1{,}14 kg/m^3, \rho_{NO_x} = 1{,}45 kg/m^3 \quad (A.9)$$

In the end the $C_{adm,CO}$ is equal to 0,0798 g/m^3 and $C_{adm,NOx}$ is equal to 0,00145 g/m^3. Finally in the case of visibility, the limit given about this problem is connected once again with the working operation and in particular it is expressed through one quantity which takes the name of *"extinction coefficient"* and in this case it is equal to K=0,005 m^{-1}. In the end, the final results obtained about the mass flow rate of fresh air are the following one.

	Flow of air [m^3/h]	**Flow of air [m^3/s]**
CO	17808,41	4,95
NOx	456274,73	126,74
Opacity	289931,75	80,54

Table A.7: Final amount of fresh air necessary to get a safe environment.

At the end of this, it is possible to define the velocity that fans have to maintain inside of the tunnel along the axis direction to eject all the noxious gases considering the semi-transverse scheme of Frejus ventilation system.

$$v = \frac{Q}{\rho A} = 2{,}70 m/s \quad (A.10)$$

This result is obtained after the computation of the flow of fresh air required, the knowledge of the mean cross section of 46 m^2 and the calculation of the density of the air considering that the pressure is obtained through the relation:

$$P = 0{,}9877^{(m/100)} = 0{,}86\ bar \quad (A.11)$$

Where *m* stands for the height at which the pressure is computed so the height of the tunnel which is equal to 12868 m. Finally the environmental density and temperatures are defined respectively as $\rho = 1{,}021\ kg/m^3$ and $T = 17°C$.

References

[1] Stéphane Colin, Heat Transfer and Fluid Flow in Minichannels and Microchannels (Second Edition), 2014

[2] David J. Benson, Computational methods in Lagrangian and Eulerian Hydrocodes, Department of AMES 0411, University of California, San Diego, La Jolla, USA, 235-241

[3] J.-Y. Chemin, Fluides Parfaits incompressibles, Société Mathématique De France,1995, 3-9

[4] A.B Metzner, Yoram Cohen, C. Rangel-Nafaile, Inhomogeneous flows of non-Newtonian fluids: generation of spatial concentration gradients, Elsevier Scientific Publishing Company, Amsterdam-printed in The Netherlands, August 8, 1978

[5] W. R. Dunbar, N. Lior, R. A. Gaggioli, The component equations of energy and exergy, March 1992, Vol. 114, 75-79

[6] K. McGrattan, B. Klein, S. Hostikka, J. Floyd, Fire Dynamic Simulator (Version 5) User's Guide, National Institute of Standards and Technology Special Publication 1019-5, October 2007

[7] K. McGrattan, S. Hostikka, J. Floyd, R. McDermott, M. Vanella, Fire Dynamic Simulator (Version 6) User's Guide, National Institute of Standards and Technology Special Publication 1019-Sixth Edition, August 21, 2020

[8] O. Reynolds, F.R.S., Motion of water and of the law of resistance in parallel channels, March 15, 1883

[9] R. Eche, The Turbulence Problem, Los Alamos Science, Number 29, 2005

[10] R.T. Cerbus, W.I. Glodburg, Information theory demonstration of the Richardson cascade, February 27, 2016

[11] Paul E. Dimotakis, Annu. Rev. Fluid Mech., Annual Reviews, Graduate Aeronautical Laboraties, California Institute of Technology, Pasadena, California, 329-333

[12] L.C.F. Andrade, J.A. Petronilio, C.E.A. Maneschy, D.O.A. Cruz, The Carreau-Yasuda Fluids: a Skin Friction Equation for Turbulent Flow in Pipes and Kolmogorov Dissipative Scales, Universidade Federal do Parà, Vol XXIX, No.2, April-June, 2007

[13] Y. Kaneda, J. Yoshino, T. Ishihara, Examination of Kolmogorov's 4/5 Law by High-Resolution Direct Numerical Simulation of Turbulence, Journal of the Physical Society of Japan, Vol. 77, No. 6, June 2008

[14] Cedric Corre et al., Direct numerical simulation of the motion of particles larger than the Kolmogorov scale in a homogeneous isotropic turbulence, ASME, Jacksonville, Florida, August 10-14

[15] T. Höhne, J.-P. Mehlhoop, Validation of closure models for interfacial drag and turbulence in numerical simulations of horizontal stratified gas-liquid flows, Elsevier Ltd., Dresden, Germany, February 2014

[16] Aaron j. Fillo et al, A fast, low-memory, and stable algorithm for implementing multicomponent transport in direct numerical simulations, Elsevier Inc., Pasadena, California, December 2019

[17] H. Homann, J. Dreher, R. Grauer, Impact of the floating-point precision and interpolation scheme on the results of DNS of turbulence by pseudo-spectral codes, Elsevier B.V., June 2007

[18] Gary N. Coleman, Richard D. Sandberg, A primer on Direct Numerical Simulation of Turbulence- Methods, Procedures and Guidelines, University of Southampton, March 2010, 1-8

[19] M. Shirzadi, P.A. Mirzaei, Y. Tominaga, RANS model calibration using stochastic optimisation for accuracy improvement of urban airflow CFD modelling, Elsevier Ltd., Niigata, Japan, August 2020

[20] D.Z. Zhang, A. Prosperetti, Momentum and energy equations for disperse two-phase flows and their closure for dilute suspensions, Elsevier Ltd., Great Britain, October 1996, 431-438

[21] P. Kumar, M.Schmelzer, R. Dwight, Stochastic turbulence modelling in RANS simulations via multilevel Monte Carlo, Elsevier Ltd., Netherlands, February 2020

[22] Paul A. Durbin, Turbulence closure models for computational fluid dynamics, Digital repository, Iowa state University, February 2018

[23] Olivier Darrigol, Joseph Boussinesq's legacy in fluid mechanics, Elsevier Ltd., Paris, June 2017

[24] Tuncer Cebeci, analysis of turbulent flows with computer programs, Elsevier Ltd., Oxford, 2013, 90-99

[25] A. Abbà, C. Cercignani, L. Valdettaro, Modelli e simulazioni LES di correnti turbolente e applicazione alla combustione, Politecnico di Milano

[26] H. Hattori et al., DNS, LES and RANS of turbulent heat transfer in boundary layer with suddenly changing wall thermal conditions, Elsevier Inc., Nagoya Institute, Japan, May 2013

[27] Promat, Tunnel fire protection, July 2008, 1-11

[28] Inland transport committee, Recommendations of the group of experts on safety in road tunnels, Economic and social council, December 2001

[29] Lamberto Mazziotti, Gli incendi in galleria:un problema anche per le squadre di soccorso, Vigili del fuoco: soccorrere in sicurezza, Roma Aprile 2002

[30] World road association (PIARC), Road tunnels: vehicle emissions and air demand for ventilation, France, 2019

[31] Fabrizio Bonomo, Valichi autostradali, Kineo 15, Il traforo del Fréjus, 1998

[32] Massimo Chiarelli, La ventilazione meccanica connessa alla sicurezza delle gallerie stradali, Ingenio

[33] World road association (PIARC), Longitudinal ventilation, France, 48-57

[34] H. Sacht, L. Braganca, M. Almeida, R. Caram, Study of natural ventilation in wind tunnels and influence of the position of ventilation modules and types of grids on a modular façade system, Elsevier Ltd., Tallin, October 2016

[35] M. Patrcucco, F. Paolo, M. Luisa, M.G. Pregnolato, Sistemi di ventilazione nelle attività cantieristiche in sotterraneo: aspetti di sicurezza e salute del lavoro, Politecnico di Torino

[36] Lombardi SA, Traforo autostradale del Frèjus: galleria di sicurezza, Minusio 2006

[37] P. Persico, T. Zuccaro, Valutazione del rischio incendio nei tunnel autostradali, Ingenio

[38] World road association (PIARC), Manuale delle gallerie stradali, France, 2015

[39] C.C. Hwang, J.C. Edwards, The critical ventilation velocity in tunnel fire a computer simulation, Pittsburgh, USA, Elsevier Ltd., 1-19

[40] P. Cialdini, La sicurezza e la vigilanza nei trafori transalpine: Monte Bianco, Frèjus, Gran San Bernardo

[41] Sitaf Spa, Traforo autostradale Frejus – Dati traffico 2019

[42] C. Bernagaud, J.-F. Burkhart, B. Vidal, M Yaghzar, The treatment of air in road tunnels, Tunnel study centre (CETU), Bron, France, December 2016

[43] Protezione civile, Piano di soccorso binazionale – Traforo Autostradale del Frèjus, Torino, May 2014, Allegato 9.4

[44] Y. Wu, M.Z.A. Bakar, Control of smoke flow in tunnel fires using longitudinal ventilation systems – a study of the critical velocity, Elsevier Ltd., Sheffield, UK, May 2000

[45] I. Vermesi, G. Rein, F. Colella, M. Valkvist, G. Jomaas, Reducing the computational requirements for simulating tunnel fires by combining multiscale modelling and multiple processor calculation, Elsevier Ltd., London, UK, February 2017

[46] F. Colella, G. Rein, V. Verda, R. Borchiellini, Multiscale modelling of transient flows from fire and ventilation in long tunnels, Elsevier Ltd., UK, Italy, July 2011

[47] M. Pachera, X. Deckers, T. Beji, Capabilities and limitations of the fire dynamic simulator in the simulation of the tunnel fires with a multiscale approach, IOP Publishing, Ghent, Belgium, 2018

[48] F. Colella, G. Rein, L. Torero, R. Borchiellini, A novel multiscale methodology for simulating tunnel ventilation flows during fires, UK, Italy, Springer Science, February 2010

[49] A. Molina, R.W. Schefer, W.G. Houf, Radiative fraction and optical thickness in large-scale hydrogen-jet fans, Elsevier Inc., Livermore, USA, 2006

[50] D. Groen et al., Flexible composition and execution of high performance, high fidelity multiscale biomedical simulations, Royal Society Publishing, UK, 2013, 1-4

[51] F. Feyel, J.-L. Chaboche, FE multiscale approach for modelling the elastoviscoplastic behaviour of long fibre SiC/Ti composite materials, Elsevier Science S.A., Chatillon, France, April 1998

[52] S. Cosentino, Innovative modelling approaches for the design, operation and control of complex energy systems with application to underground infrastructures, ScuDo, Politecnico di Torino, Italy, 2017

[53] L. Delpopolo Carciopolo, L. Formaggia, A. Scotti, H. Hajibeygi, Conservative multirate multiscale simulation of multiphase flow in heterogeneous porous media, Elsevier Inc., Italy, Netherlands, November 2019

[54] R. Carvel, Design fires for tunnel water mist suppression systems, Stockholm, Sweden, March 2008

[55] R. Borchiellini, V. Verda, A fuzzy logic procedure for ventilation control in case of fire in long tunnels, Elsevier Ltd., Politecnico di Torino, Italy, February 2009