3D Simulation of a fire inside a river tunnel and optimisation of the ventilation using Computational Fluid Dynamics

Authors: B. TRUCHOT, A. TRIPATHI
Fluidyn-France

Abstract

CFD simulation is used to predict 3D flow inside different configurations. Tunnel ventilation system is one example of CFD application field. Ventilation system is important in all kind of tunnels, road tunnel, rail tunnel or waterway tunnel. In all case CFD is able to predict chronic pollutant dispersion or accidental consequences. This paper presents the model of a waterway tunnel in France in which the ventilation system is particularly complex, performed by the CFD software, fluidyn-VENTUNNEL. A first simulation deals with sanitary ventilation in the course of a typical scenario of 3 passenger boats in the tunnel. The second part of the paper presents the emergency ventilation scenario in case of an accidental fire on one of the boats. In that case, as soon as the detection (based on CO levels) has occurred, a PUSH/PULL system is simulated to prevent from inrush of smoke in the tunnel. Although all parameters have been chosen in order to be as penalizing as possible, results show that ventilation system is correctly dimensioned. This example case will demonstrate the interest of CFD in tunnel ventilation characterization.

Introduction

Computational Fluid Dynamics (CFD) can be a very powerful tool to prevent environmental accidents and avoid their lethal consequences. In tunnel ventilation characterization for road, rail or waterway tunnels particularly, CFD can be used to model air flows, temperature stratification, species dispersion such as chronic traffic pollution dispersion or accidental configuration (fumes or lethal species dispersion in case of fire, terrorist attack or other kind of accident) and visibility. It can help to conceive an efficient extraction system and coordinate the emergency procedures and measures.

Tragic events, these past years, have brought attention the need for a precise assessment of risk management and environment management in tunnels. In most long tunnels, a sanitary ventilation is required to bring fresh air and dilute pollutants generated by the traffic. This ventilation can be “natural” by opening of stacks, or “mechanical” with the addition of longitudinal accelerators or vertical extractors. The purpose that CFD can fulfill then is to check whether the proposed ventilation is enough to ensure that the pollutants levels stay in acceptable concentrations even in adverse conditions of weather (wind blowing the other way) or traffic (in advent of a traffic jam). In that case, the populations near the heads of the tunnels have to be taken into consideration as well.

The second item on which CFD can help is the simulation of crisis management in terms of ventilation in case of a fire in the tunnel or networks of tunnels. In that case, the simulation will model the fire by adding source terms for mass, buoyancy and heat and will modify the flow in the tunnel accordingly. The output that can be analysed from the simulation are the time needed for captors located along the tunnel walls to register the presence of pollutant by-products of fire, such as CO, the time and the possibility for the fan to get into emergency mode (maybe in reverse mode), and then the level of pollutants in the tunnel, the smoke extraction, the temperature stratification and the time available for evacuation of peoples trapped in the tunnel.

The case presented here to illustrate possibilities of CFD is the simulation of the ventilation in a waterway tunnel in France, first in sanitary mode and then in emergency mode. The ventilation of this tunnel is particularly complex with 19 evacuation chimneys, each of them include an extractor. The first case considers chronic emission (exhaust gas from boats) and ensures that limit concentration of pollutant is not reached. The second case is a fire case. Fire of one boat is considered. In such a case, extractors in chimneys are remote-controlled by CO captors that have been numerically considered. For this second case, the aim of the simulation is to show that gases emitted from combustion are evacuated to ensure security of the other boat.

Geometry and mesh

The tunnel is a one-lane tunnel constructed in order for a canal to cross a hill. The tunnel is 3 km long and 6 m wide. The height of the tunnel is divided into 2.6 m of water and 3.6 m of air. A schematic representation of the tunnel is presented on Figure 1. This picture is a longitudinal section of the tunnel. The ventilation system is composed of 19 evacuation stacks but 3 of them were abandoned and 2 have been closed. Those 5 closed chimneys were plotted in grey on the figure. All the chimneys have their own dimensions, section and height. On figure 1, barges can only go from left to right.

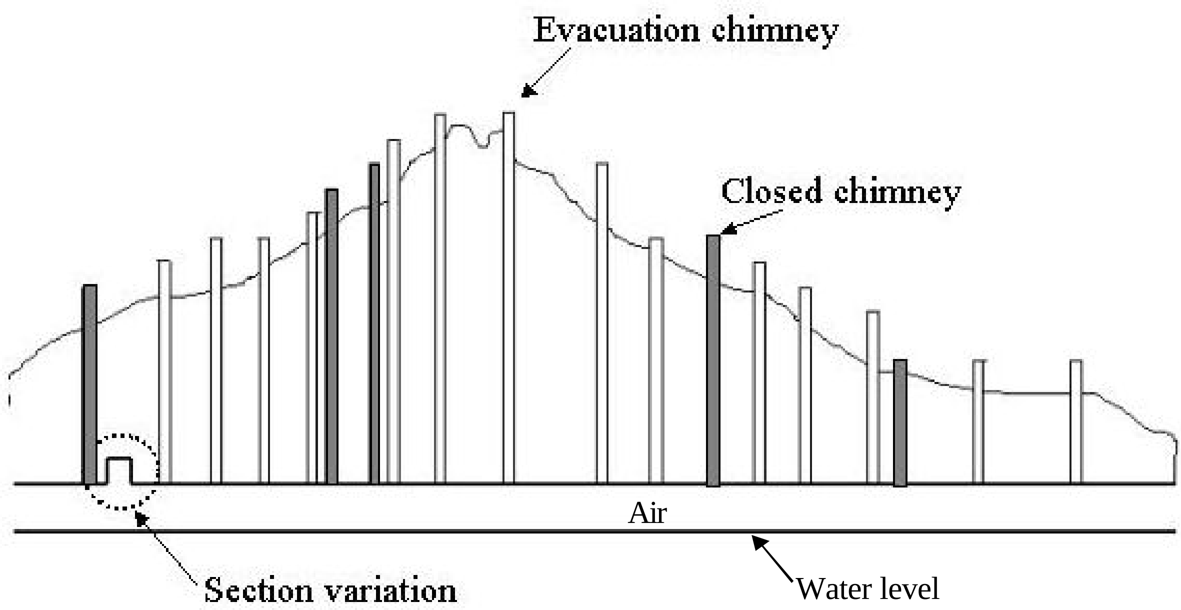


Figure 1: Schematic longitudinal section of the tunnel.

Extractors are located inside each chimney one meter below the outlet. The extractors can blow in both directions and are remotely controlled using captors. Therefore the direction can be modified in function of concentration of CO inside the tunnel. This specific capability will be detailed in the section dealing with the fire case simulation. The picture on Figure 1 also shows the section variation located in the tunnel that implies flow perturbations, mainly when barges go through the tunnel.

The geometry and the mesh created contain the stacks, the barge (see definition of scenario below) and tunnel. The bottom of the geometry is considered to be the water level. Figure 2 shows different views of the mesh. The mesh is structured for a better precision and made of 511 472 cells. As shown on the figure, the mesh is refined near chimneys to model the flow with high precision in these zones. It is a non-conform mesh to accommodate fast transitions without too many cells and it is a moving mesh to take into account the movement of barges inside the tunnel. One barge is 38.5 m long, 3 m high and 5.05 m wide. The section of the barge is presented on picture (c) of figure 2.

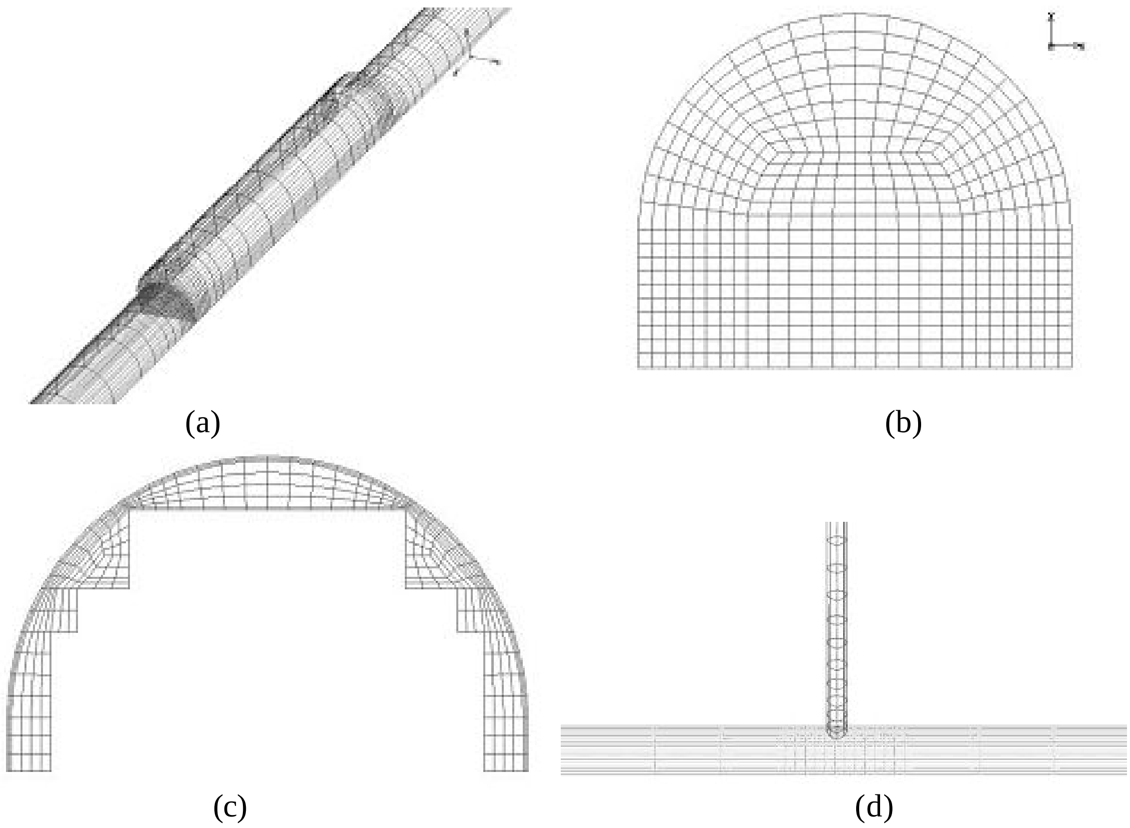


Figure 2: Mesh views.

(a): Isometric view of the mesh near the section variation ; (b): Section of the tunnel ; (c): Section of the tunnel containing one barge ; (d): mesh near one chimney.

Sanitary scenario

The first scenario modelled here is a chronic dispersion case. Because the tunnel can be used by all type of boats and barges with passengers, it is important to ensure that pollutant concentration in a chronic situation is low enough below the environmental standards in all situations. Therefore the maximum allowable number of barges of two inside the tunnel has been considered as well as the minimum time between them (10 minutes).

The CFD simulation was performed using *fluidyn*-VENTUNNEL, a module of the fluidyn-*fluidyn*-MP software developed by Transoft International, dedicated to CFD simulation in network of tunnels. The flow modelling is based on the 3D compressible fluid mechanic equations: mass transport, Navier-Stokes equations and energy conservation. Mass equation is written following a multi-species approach, that is to say mass fractions are solved for each species and density is computed from those mass fractions. Source terms are added in those transport equations to account for the species emissions from sources, chronic or accidental. Turbulent terms are modelled following a RANS approach with the standard k-ε turbulence model. The boundary layer is accounted for by setting up a wall roughness. A 3rd order convection scheme has been used with a 6-step Runge-Kutta integration to solve for unsteady flow for this case.

A first simulation without the emission is performed to obtain a steady state that will provide the initial conditions for a dispersion simulation. For such a simulation, wind external conditions are important because the pressure difference at the outlets due to the wind can induced important flow in the domain. In the particular case of the study reported here, different wind configurations have been considered. The most penalizing cases (wind blowing against the fans) as well as the most frequent cases (based on the wind rose of the site) have been selected for study.

The results of the steady-state simulation are presented at specific locations on Figure 3.

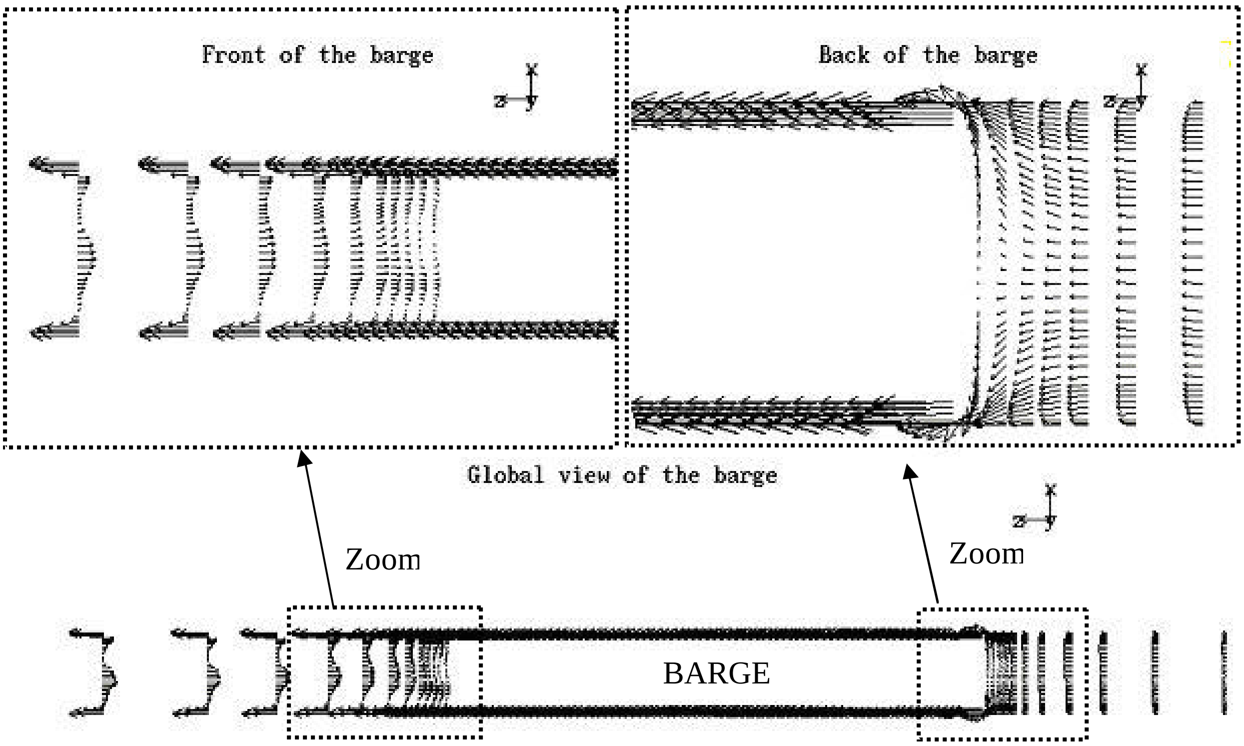


Figure 3: Velocity vectors around one barge in the tunnel in an horizontal section.

Figure 3 shows the particular shape of the flow around a barge. The flow is accelerated, due to the section reduction on each side of the barge and recirculation zones can be observed in its wake. This flow configuration is due to the ventilation system and to the section of the barge, which is almost as large as the tunnel section.

On the initial conditions provided by the steady-state simulation, the emission of pollutant is added. The emission is located at the exhaust of the barges. The pollutant monitored here is NO_2. The emission rate of NO_2 from the barge exhauster is $11.51*10^{-6}$ kg/s, assuming that all NOx, out of the combustion process of the engine, has been converted to NO_2, which is a penalizing assumption.

The concentration levels of NO_2 at the back of both barges are plotted on Figure 4. In this case, dispersion is always dominated by convection and not by diffusion phenomena because of the high speed of flow around barges. For the case plotted on Figure 4, the difference of concentration behind the two barges is due to the flow through each barge: In the picture on the left, flow goes from head to back of the barge while in the picture on the left, flow from back to head of the barge:

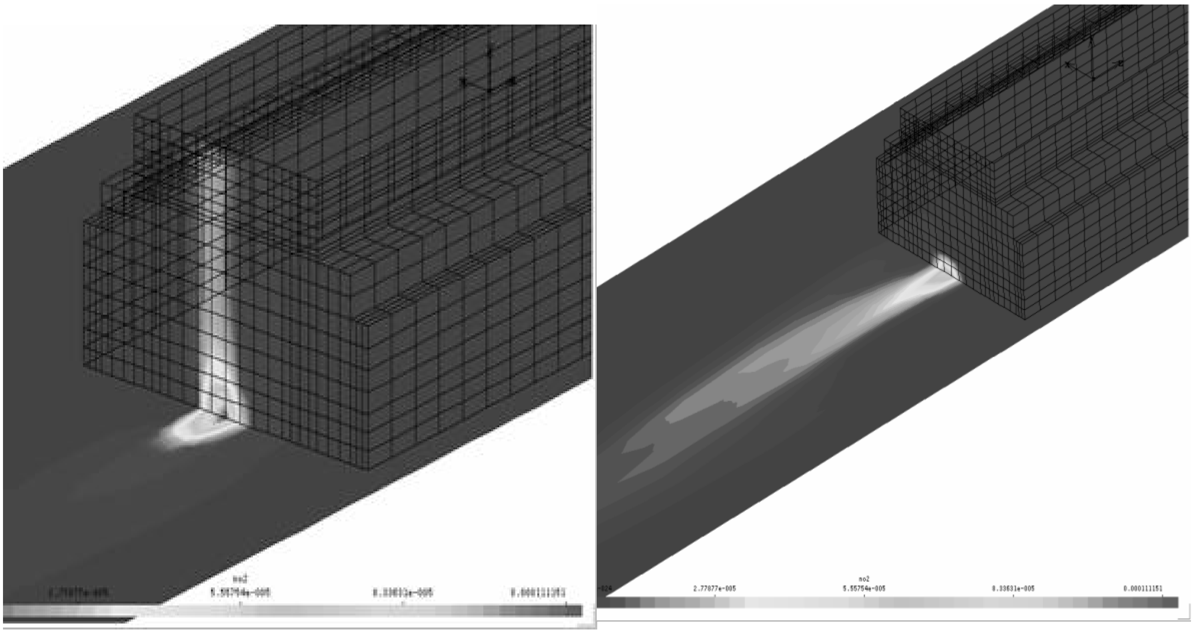


Figure 4: Concentration of NO_2 behind each barge in an horizontal section located at the water surface.

Iso-concentration surfaces are plotted in Figure 5 and Figure 6 : inside the volume covered by the surface, the pollutant concentration is above the value at which the surface was plotted. The pollutant concentration threshold for a safe environment is 0.4 ppm. The results show that the cloud of NO_2 with concentrations above 0.4ppm stays very close to the exhaust and doesn't affect the passengers in neither of the two barges. So from the point of view of the health concerns, the ventilation system works correctly as pollutants remain in a very local area.

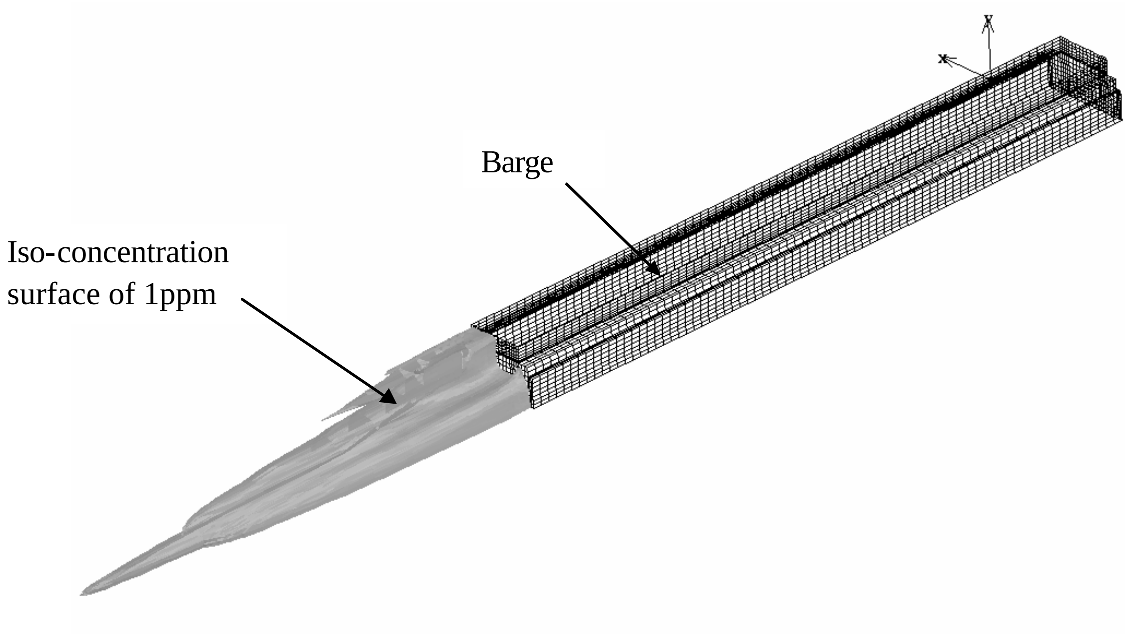


Figure 5: Iso-concentration surface of 1 ppm

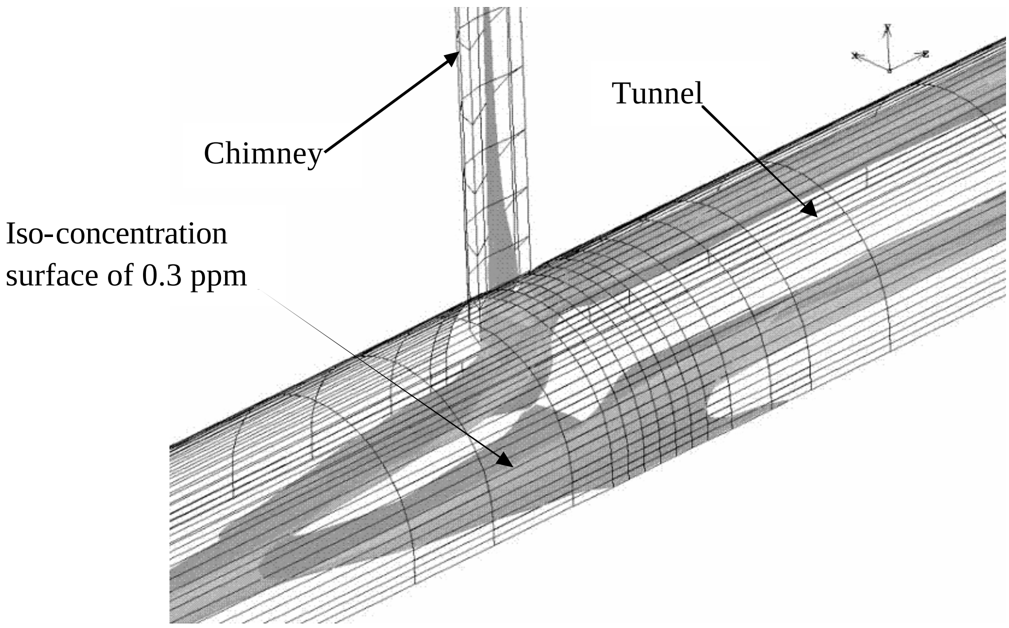


Figure 6 : Iso-concentration surface of 0.3ppm next to one of the chimneys

Fire scenario

The second group of simulations deals with a fire on board of a barge. For such a case, normalised emission curves provided by CETU (Centre d'Etudes des Tunnels) for cars and lorries inside road tunnels are interpolated to obtain energy and pollutant emission curves for a barge. The two fire scenarios investigated here are: one scenario for a fire on board of a regular passenger boat and one scenario for a fire on board of a barge full of flammable products (wood, cereals, etc.). The powers considered for each fire are respectively 30 MW and 100 MW. The power curve used in this study to model the fire of the barge is plotted on Figure 7.

Pollutant species emission are directly linked to the power of the fire. According to CETU, the CO_2 production is 0.1 $kg.s^{-1}$ for 1MW and CO production is given by the ration $[CO_2]/[CO]=5$, corresponding to a penalising case for a under ventilated fire. Concentration of CO must always be lower than 150 ppm, which is the safety threshold for a 120 minutes exposure (first effect on human beings [1]), everywhere in the tunnel.

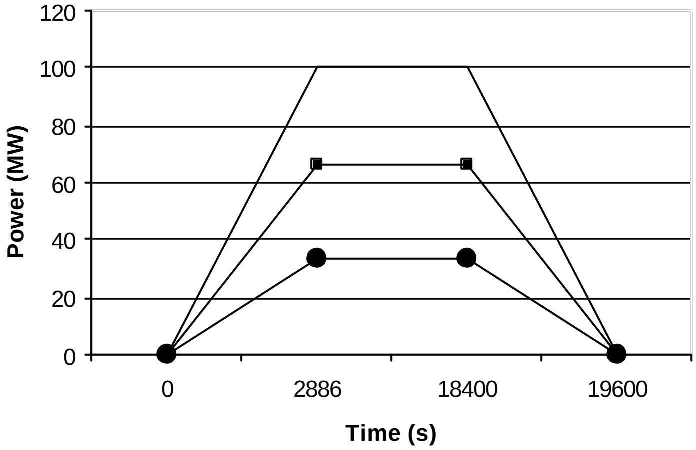


Figure 7: Fire power distribution against time for a flyboat.

: total power; ■ : convected power; ●: radiated power.

Captors are distributed every 100 m all along the tunnel to measure concentration of CO. The concentration limit of 150 ppm will trigger the emergency extraction system. Once concentration limit is reached for one of the captors, ventilation system will start working in PUSH-PULL mode: extractors directly around the sides of the barge in fire will pull air, extractors around the first ones will stop, and extractors on the left side of the tunnel will push air and extractors on the right side of the tunnel will pull air.

Monitor points were introduced in the code to simulate CO captor behaviour. This numerical strategy enables to simulate the PUSH-PULL ventilation system. Evolution in time of CO concentration for one of the monitor point that detects the fire in one of the cases studied is shown on Figure 8. From the instant the fire is detected, ventilators will take 120 sec to start working in PUSH-PULL mode. This delay must be added to the fire detection delay, which is function of fire position and power and of the flow. In the worst case, fire detection delay can reached 430 s, that is to say approximatively 7 mn. This delay can appear to be very long, however detection system is sufficient to ensure passengers safety in terms of pollutants concentration in the tunnel. CFD is also able to help to determine a better repartition of captors if required, that was not the case for those simulations. For the example curve presented on figure 8, the fire detection delay is 75 s.

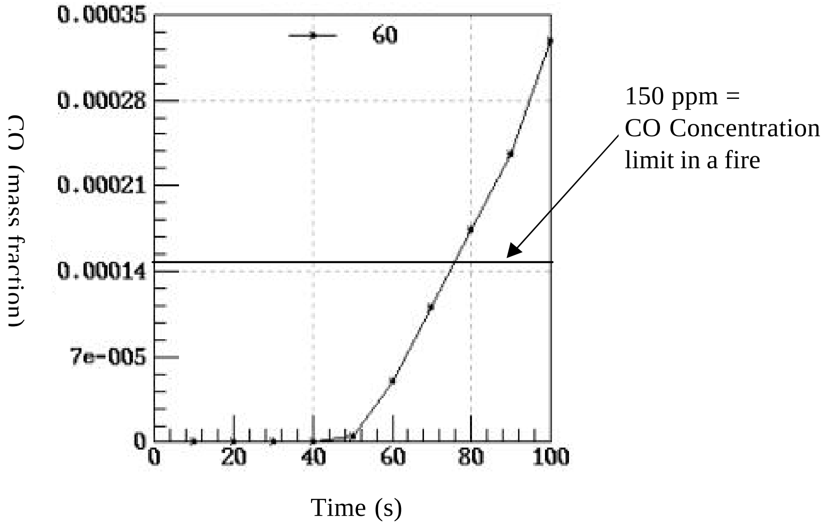
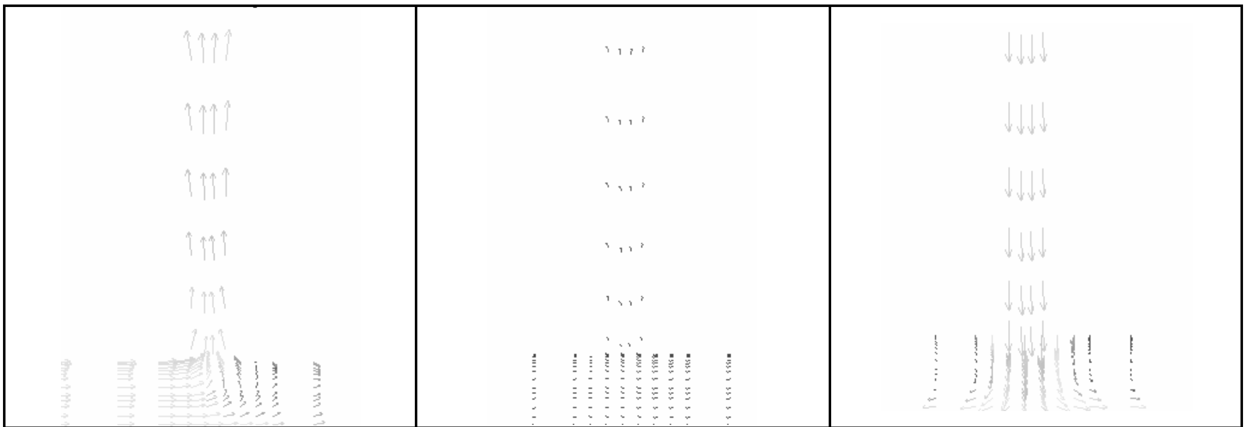


Figure 8: Evolution in time of CO concentration for one numerical captor.

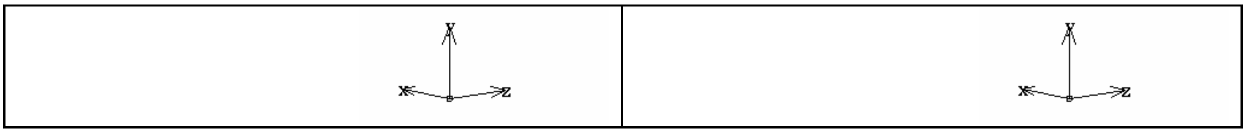
Initial conditions concerning air composition and flow inside the tunnel of this second group of simulations are taken from steady conditions for the sanitary study in order to model with greatest fidelity the reality inside the tunnel

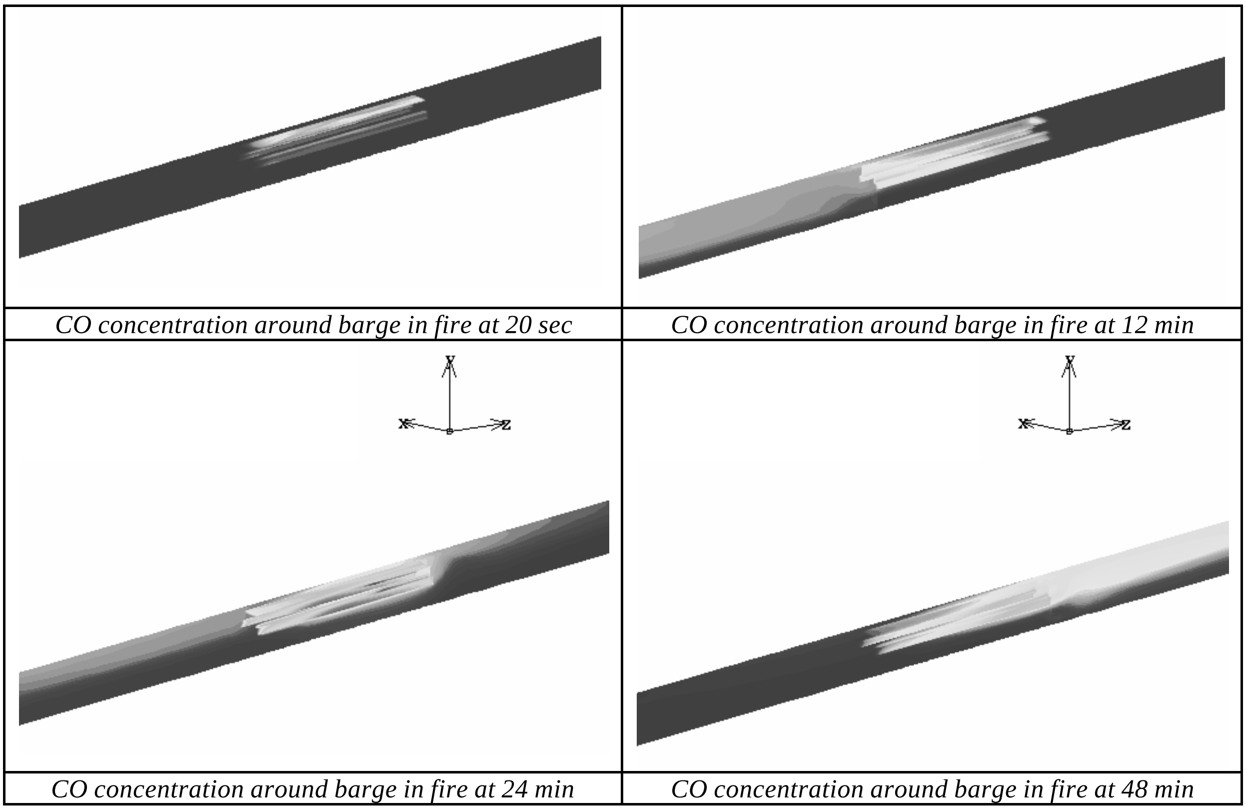
Although extractors of chimneys besides the barge in fire are stopped once the fire is detected, there will be a flow through them due to natural convection. However, this flow is not displaying a steady behaviour from the beginning. Its direction changes in time inside the chimneys as shown on Figure 9:



Flow through chimney 2 at 20 secs	*Flow through chimney 2 at 12 min*	*Flow through chimney 2 at 48 min*

Figure 9: Evolution in time of flow through chimney stopped





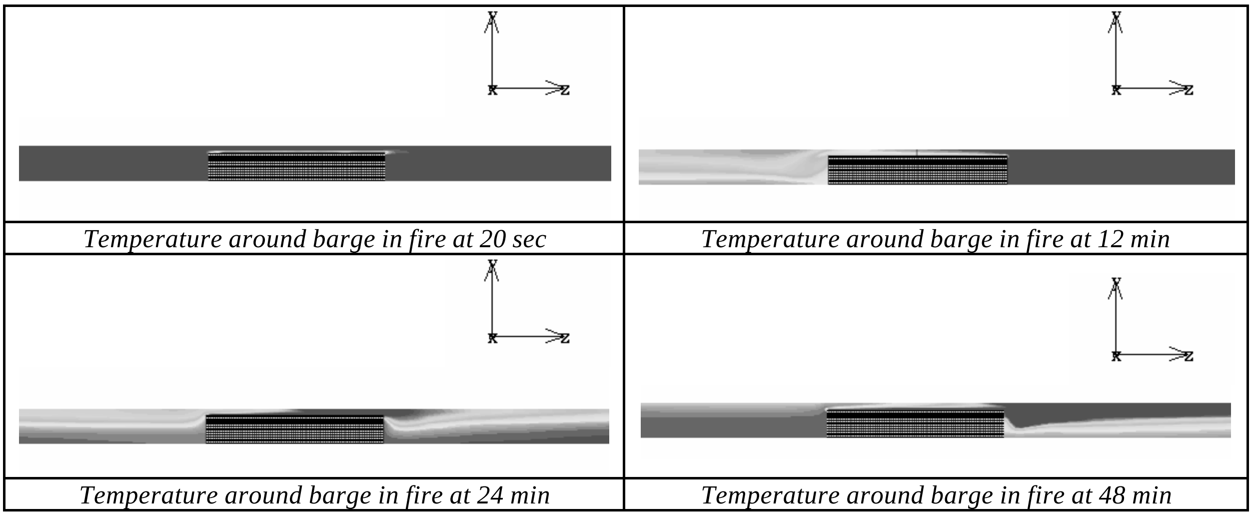
CO concentration around barge in fire at 20 sec | *CO concentration around barge in fire at 12 min*

CO concentration around barge in fire at 24 min | *CO concentration around barge in fire at 48 min*

Figure 10: Evolution in time of CO concentration around barge in fire

As NO_2 concentrations were representative of the sanitary study, CO concentration evolution is analysed here. The following picture shows that CO concentration around barge in fire is not steady in time and needs time to stabilise because of the unsteady behaviour of the flow inside the tunnel caused by the PUSH-PULL system. Once the flow is steadied, after about 48 min, the CO cloud remains at the right side of the barge without affecting the other barge. So ventilation system is considered to be satisfactory.

Temperature distribution was also studied in order to determine if high temperature zone could reach passengers. Maximal temperatures are reached in same zones of CO maximal concentration and follow the same evolution, as plotted on Figure 11:



Temperature around barge in fire at 20 sec | *Temperature around barge in fire at 12 min*

Temperature around barge in fire at 24 min | *Temperature around barge in fire at 48 min*

Figure 11: Evolution in time of temperature around the barge in fire

The temperature distribution in the tunnel and in chimneys is also very important because the temperature governs extractor working. Extractors for this tunnel were dimensioned to work under a temperature of 200°C during 120 mn that is to say that if the temperature is higher than 200°C during 120 mn, extractors break down. This condition is never reached in the simulations here.

Simulations have allowed to determine if a predefined ventilation system was quite appropriate to carry out exigencies of health and to keep the tunnel as secure as possible in case of fire. After its validation in both of the situations, the power of each extractor for the most unfavourable situation was evaluated by taking account total pressure loss from the inlet of the flow by both of the heads of the tunnel to the outlet by the chimneys.

Conclusion

In this paper, one CFD application case in tunnel was shown. This case deals with waterway tunnel and considered not only a sanitary case but also an accidental situation. In the first case, CFD ensures the capability of the ventilation system in standard mode to evacuate exhaust gas that guarantees an acceptable atmosphere inside the tunnel for passengers of tourism boat. In the second case, the fire barge simulations use the emergency ventilation system with PUSH/PULL system that enables to evacuate gas quickly. Simulation has shown that in case of fire, the other boat in the tunnel will not be inside the panache of fumes from the fire barge thanks to the PUSH/PULL system.
The other results inferred from simulations are: the temperature reached inside the tunnel, the expected visibility for fire-fighters operation and passenger evacuation, the optimisation of captors location, the temperature reached for equipments such as extractors.
This example demonstrates the interest of CFD for ventilation system characterization in tunnels to predict physical behaviour in different cases with a great precision. Numerous other aspects can be considered numerically such as vehicles displacement, ventilation system control with captors, external wind conditions or traffic, etc.

References

[1] Ventilation, Les dossiers techniques du CETU, November 2003.