AIVC
#13,777

3D Simulation of Dynamic Barriers Against Fume and Gaseous Toxic Substances

Cumo F[1], *Lizio G*[1], *Rossetti S*[2]
[1] Dip. Fisica Tecnica, Università di Roma "La Sapienza", Italy
[2] Centro di Conservazione Opere d'Arte –CNR -Roma, Italy

Abstract

The paper deals with a three dimensional modelling of an experimental device realised in the Department of "Fisica Tecnica" of the University of Rome "La Sapienza" in co-operation with the Italian National Council of Researchers (CNR) in order to investigate the performances of shutter type air curtains as a dynamic barrier against the diffusion of fumes or airborne toxic substances. This kind of dynamic barrier could be used as a new element for the compartimentation in fire-protection systems and also for protection against the diffusion of noxious gaseous releases following to accidents in chemical processes. Actually air curtains main applications in safety industrial devices are limited to the segregation of dangerous gaseous sources and to the conveying of pollutants toward exhaust systems. Shutter type single sided downward flow air curtains realise a Push and Pull System: the push opening are sided as the pull system; the downward air can be directed toward the incoming out side air at an angle ranging from 0° to 15° with respect to the vertical. The goal of the paper is to investigate if those shutter type air curtains are rationally designed and to evaluate their effectiveness in containing pollutant sources in defined spaces, starting with the design aspects deeply studied in last ten years (1, 2, 3). In the following the main design parameters of those dynamic barriers will be considered in order to evaluate their influence on the air curtains performances and to redefine current design criteria.According to the flow ratio method (1), these parameters are :width of the push and pull openings, length of the push and pull flanges and push air flow rate

A numerical approach based on a CFD code (Fluent 3.2) has been used in the first part of the research program; the second part will be the realisation of the experimental device with a design based on the results of the fluid dynamic simulation.

The numerical simulation of the airflow velocity and temperature and of the CO_2 concentration has been carried out using the CFD code Fluent. This is three dimensional, finite volume elements, able to model full multi-components species transport and the diffusion of species due to temperature gradients (Soret diffusion) using the $\kappa - \varepsilon$ turbulent model (two equations model (4)). The general expression for the diffusion mass flux vector of species i' in full multi-components diffusion is given by :

$$J_{i'} = \frac{C^2}{\rho} \sum_{j'-i,j'\neq i'}^{n} M_{i'} M_{j'} D_{i'j'} d_{i'} - D_{i'}^{T} \frac{\nabla T}{T} \qquad (1)$$

where:

$C = molar\ concentration\ of\ the\ mixture$

$M_{i'} = molecular\ weight\ of\ species\ i'$

$D_{i'j'} = multicomponent\ diffusion\ coefficient\ for\ species\ pair\ i' - j'$

$d_{j'} = mechanical\ diffusion\ vector\ for\ species\ j'$

$X_{j'} = mole\ fraction\ of\ species\ in\ the\ mixture$

$D_{i'}^{T} = thermal\ mass\ diffusion\ coefficient\ for\ species\ i'\ in\ the\ mixture$

The first term of the second member of eq.(1) is the contribution of concentration gradients to the diffusion mass flux vector and the second term represents the Soret effect, that is the mass transfer due to temperature gradients. This general form of the diffusion flux appears as a modelling option. When the fluid mixture is assumed to behaves an ideal gas, equation (1) can be expressed in terms of mass fractions and, with some mathematical manipulations, be reduced to yield:

$$j_{i'} = \rho \frac{M_{i'}}{} \sum_{j'=1, j' \neq i'}^{n} D_{i'j'} \left(\nabla mj' + mj' \frac{\nabla M}{M} \right) - D_{i'}^{T} \frac{\nabla T}{T} \qquad (2)$$

The species equation than becomes:

$$\frac{\partial \rho m_{i'}}{\partial t} + \nabla \cdot \left(\rho V m_{i'} \right) = \nabla \cdot \left[\rho \frac{M_{i'}}{M} \sum_{j'=1, j' \neq i'}^{n} D_{i'j'} \left(\nabla m_{j'} + m_{j'} \frac{\nabla M}{M} \right) \right] + \nabla \cdot \left[D_{i'}^{T} \frac{\nabla T}{T} \right] + R_{I'} \qquad (3)$$

The indoor space in the numerical simulation is schematised as a rectangular room (4x3x3 meters) with an open door (0,9 meter width) where is located the air curtain, with a large closed window on the opposite wall. The CO_2 contaminant source has been located in the central part of the room having an emitting surface of about 2 m^2 and heigth of 0,5 m. The boundary conditions for outdoor are expressed by fixing the value of the pressure (assumed equal to the atmospheric pressure) on the door and the window, and the value of the wall temperatures (mean walls temperatures 20°C). The following simulation outputs were obtained with a 3D geometry reported in Figure 1

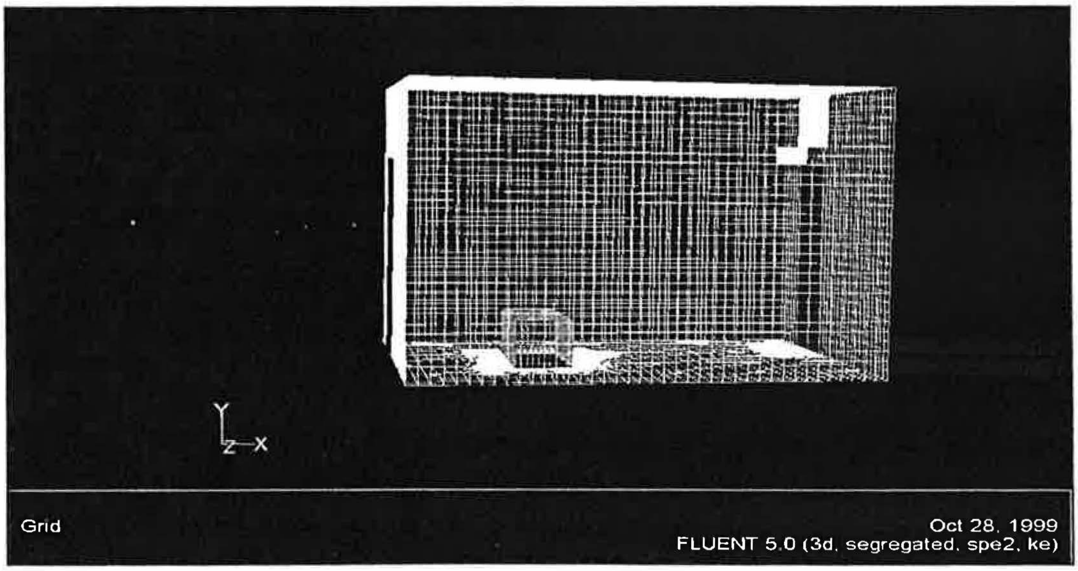
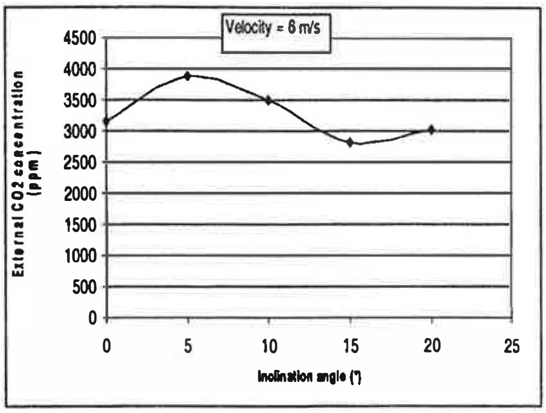


Figure 1. Room geometry and mesh

Different values of CO_2 concentration, at an height of 1,5 m, inside and outside the room, varying push angles, air curtain lengths, inclination, air velocity and flow rate, are reported in figures 2,and 3. In figures 4 is reported the CO_2 concentration versus the push air velocity, varying the flow rate.



Air curtain Length (m)	External CO_2 concentration (ppm)
0.9	2810
1.2	2200

Figure 2. CO_2 external concentration varying push air inclination angle and air

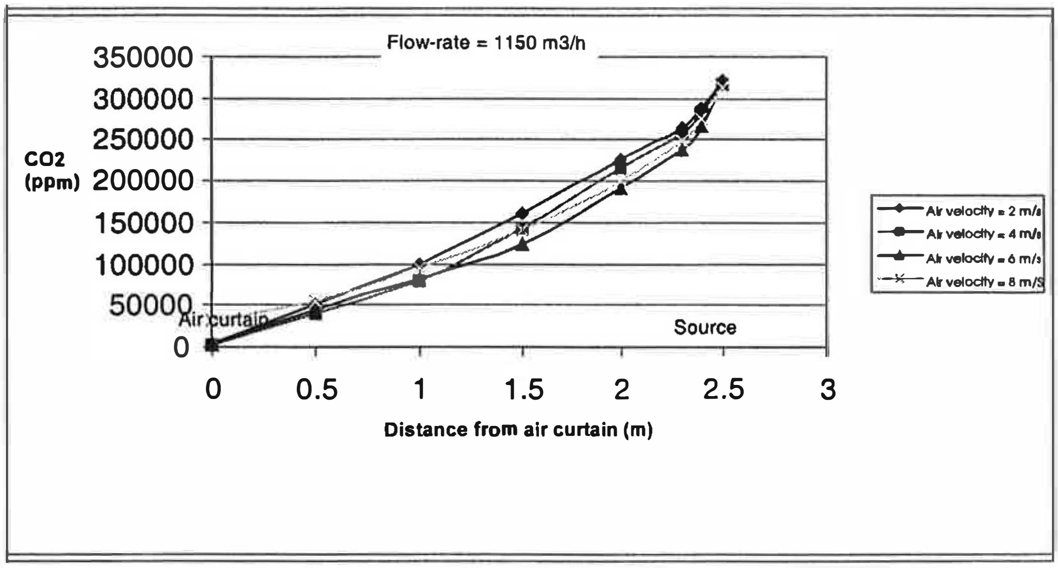


Figure 3. CO_2 concentration inside and outside room, varying push air velocity

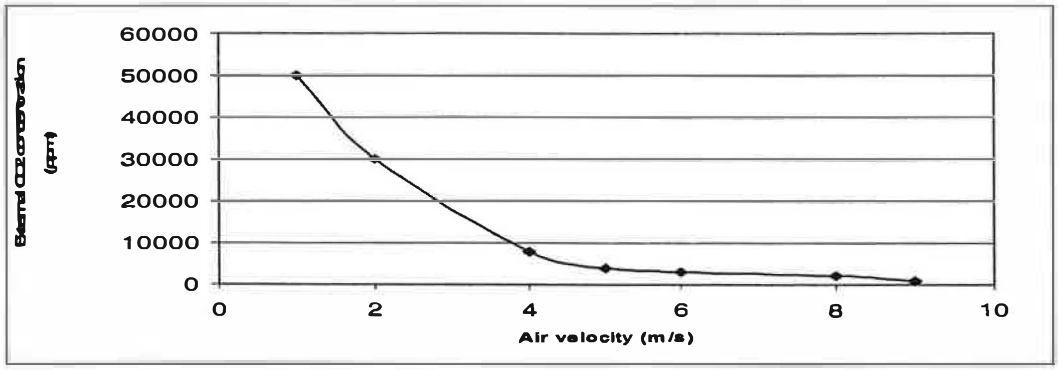


Figure 4. CO_2 concentration inside and outside room, varying push air velocity and flow-rate

In figure 5 it is shown the CO_2 concentration inside the room with the following, optimised input conditions: air velocity: 6 m/s; air curtain length:1.2 m.; flow rate:3456 m^3/h.; inclination angle:15°.

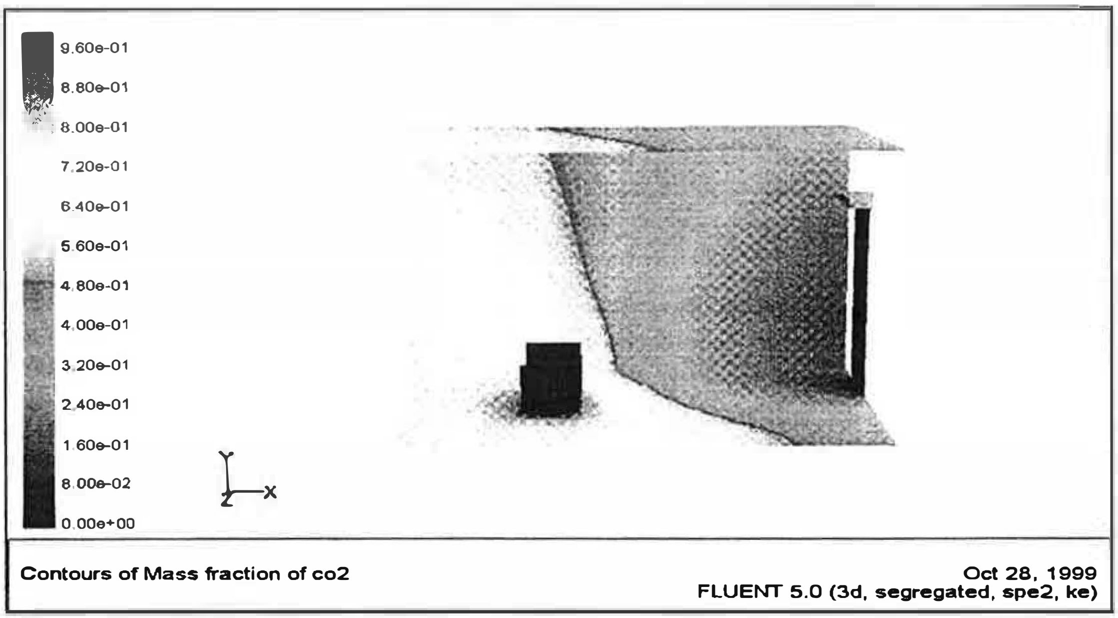


Figure 5. CO_2 concentration inside room

In conclusion analysing the numerical simulation it is possible to evince the following considerations when an air curtain is used as a dynamic barrier against the diffusion of fumes and toxic airborne substances:

- A push air velocity ranging 5-6 m/s reduces the pollutants concentration outdoor, respect to the source concentration, of about 100 times.
- The push air flow rate, independent from the indoor volume, has no great influence on the outdoor concentration of pollutants, so it is better to design the air curtain fixing the push air velocity and then adjuste the push air flow rate.
- The push air flow rate better inclination toward the room is about 15° respect to the vertical
- The optimal length of the air curtains is .longer than the opening width.

References:

1. Hayashi T, Howell R.H., Shibata M. And Tsuji K., Industrial ventilation and air conditioning, CRC Press, Boca Raton, FL 1987.
2. Heinsohn R.J., Industrial ventilation: engineering principles, John Wiley and Sons, New York, 1991.
3. Green K., Energy saving for industry, Heating and Conditioning 57 (2), 1987.
4. Patankar S.V., Numerical heat transfer and fluid flow, Hemisfere, N.Y., 1980