CHEMICAL ENGINEERING TRANSACTIONS

VOL. 26, 2012

Guest Editors: Valerio Cozzani, Eddy De Rademaeker
Copyright © 2012, AIDIC Servizi S.r.l.,
ISBN 978-88-95608-17-4; **ISSN** 1974-9791

A publication of
AIDIC
The Italian Association
of Chemical Engineering
Online at: www.aidic.it/cet

DOI: 10.3303/CET1226067

Response Surface Methodology for Analysis of an Air Curtain Used as Emergency Ventilation System in a Tunnel Fire

Felipe Vittori*[a], Luis Rojas-Solórzano[a], Michel Pavageau[b]

[a]Simón Bolívar University, Department of Energy Conversion and Transport. Caracas, Venezuela.
[b]Ecole des Mines d´Albi, Campus Jarland, 81013 ALBI, France.
vittorifelipe@yahoo.com

This paper presents results obtained by a multi-variable method combined with CFD to study the performance of an air curtain in a fire scenario inside a tunnel. An air curtain is a plane stream of air blown across an opening to create a singularity that hampers the free air movement through this opening. An air curtain is used to produce a barrier effect while permitting traffic of people, vehicles, materials or objects between the areas the air curtain separates. They are widely used at building entrances to ensure a constant inside-outside temperature difference. Air curtains are also often mounted in the front of refrigerated food counters and open-shelves for customers to easily see products without having to open a door. In many circumstances, an air curtain can be seen as an impinging plane jet. The main flow is then composed of three or four basic regions depending on the jet height H to width e ratio: a free jet (just downstream of the discharge nozzle exit), a development region (absent if H/e is small), an impingement region including a stagnation zone, and a wall-jet region. The air curtain investigated in this study is part of an emergency ventilation system designed to stop smoke propagation during fire in a tunnel. The whole system is made of 2 air curtains forming a confinement cell within which smoke should be retained. A traditional CFD study would take a lot of time and computational resources. Therefore, CFD was combined with Response Surface Methodology (RSM), a useful tool to produce results that take into consideration many variables and ranges in an optimal way. The virtual set-up investigated in this study describes a 2D tunnel of length 100 m with one confinement cell and a fire source designed from real fire simulations for testing ventilation systems. The Reynolds number based on the flow velocity and width at the discharge duct of the air curtain was varied from $4{\cdot}10^5$ to $1{\cdot}10^6$. The height-to-width (H/e) ratio was varied from 2 to 30. The efficiency of the air curtain was assessed through the temperature and smoke concentration difference between the tunnel exit and the confined section. The RSM method proved to be an efficient way to study multiple ventilation conditions for a given fire scenario from a reduced number of traditional CFD simulations. Air curtains are shown to be effective barriers able to confine heat, reduce the temperature in the tunnel outside of the confinement cell, and thus facilitate evacuation. Smoke leakage occurs through the curtain due to two main reasons: the first one is based on the development condition of the impinging jet, and the second is related to mass conservation within the confinement section. The results show that the best air barriers are those with a high Reynolds number and a small H/e ratio.

Please cite this article as: Vittori F., Rojas-Solorzano L. and Pavageau M., 2012, Response surface methodology for analysis of an air curtain used as emergency ventilation system in a tunnel fire, Chemical Engineering Transactions, 26, 399-404
DOI: 10.3303/CET1226067

1. Introduction

Mechanical ventilation systems are necessary in tunnels to ensure the presence of fresh enough air for people and engines. They are usually designed to drive fresh air inside the tunnel, and simultaneously push out the polluted air from the tunnel. In many cases, tunnel ventilation systems are designed for both normal operation and emergency cases (fire for example). A well designed emergency ventilation system should ensure a rapid extraction of toxic gases keeping safe the evacuation and rescue team access routes from both tunnel ends.

Traditional emergency ventilation systems are classified according to Kashef et al. (2003) in longitudinal, transversal and semi-transversal. Their design depends on empirical relations that not always guarantee the desired flow patterns. Therefore, preliminary test studies are required (Galdo-Vega et al., 2006) every time it is aimed to implement a ventilation system in a tunnel. Despite a certain number of limitations, CFD studies are very useful in this regard. They tend to be ever more reliable for such analyses. However, experimental verifications are sometimes unavoidable, even though they are, generally, very expensive.

Because air curtains are particularly unstable flows with a rather unpredictable behaviour that strongly depends on boundary conditions, the present numerical study was undertaken to investigate to what extent air curtains can be efficient in fire safety in tunnels.

The air curtain system of this study is based on the Gupta et al. (2007) design whereby the considered tunnel can be divided into adjacent cells by a series of parallel air curtains allowing smoke confining locally, and providing occupants with the possibility to escape the tunnel in both directions. Eventually, if the cell in which smoke is confined is full of smoke and without oxygen, the fire self-extinguishment could occur. The system was thought to also make it possible for rescue teams to access the incident place more freely from both tunnels ends.

Based on the principle of cellular confining, a parameter study of the performance of a single-cell system based on the use of 2 air curtains has been performed. Some of the dynamical and geometrical parameters of the air curtains forming the system were varied to investigate what configurations are most efficient. Relevant results are presented in this paper.

2. Geometry and boundary conditions

The study was carried out numerically in 2-D by Elicer-Cortés et al. (2009) who demonstrated that the plane jet simulation was able to reproduce the flow characteristics of the air curtain arrangement. Figure 1a shows a half tunnel with its respective dimensions; it has no slope, neither curvature. The confinement cell is formed by two curtains and one extractor. To reduce computational cost and time, the fire source was located in the middle of the tunnel, right under the extractor. This allows to establish longitudinal symmetry, and to work with half of the domain, as shown by the boundary conditions in Figure 1b.

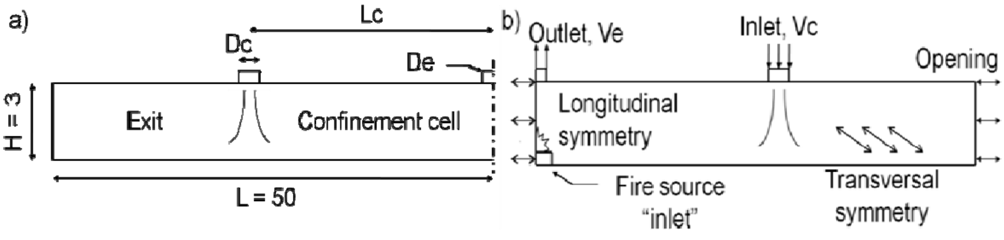

Figure 1: a) tunnel 2-D geometry and b) boundary conditions; dimensions in meters

The geometry was based on the tunnel A13 of the A86 West Underground Link-up of Paris, France, where only small vehicles transit, with no trucks or lorries. This helped to define the fire source as a volumetric source of 1 MW of heat release rate, a 5.8 m^3/s smoke flow, a 0.98 m diameter and 267 °C that represents a small vehicle on fire according to Mégret and Vauquelin (2000). Radiation was not taken into account in the simulations according to Galdo-Vega et al. (2006) and Gobeau et al. (2002)

recommendations. Those authors have indicated that radiation becomes important, and need to be modelled when the fire temperature is above 327 °C.

The ambient conditions were 25 °C and 101 kPa. The tunnel opening had a relative pressure of 0 Pa, which avoids the enforcement of velocity directions and allow the circulation of air and smoke based on the numerical solution of the flow field. Turbulence intensity at the air curtain blowing nozzle exit was set to 10 %, which represents a high level of turbulence. A mesh sensitivity analysis, based on velocity magnitude was performed until a mesh of 228722 nodes was obtained, with squares of 0.05 m. The mesh was refined locally in the jet region, from the curtain discharge until the tunnel floor with squares of 0.0125 m. Additionally, a refinement layer of 0.5 m of height composed of rectangular elements 0.0125 m-height by 0.05 m-long was built on the tunnel floor.

2.1 Governing equations

The simulation of air and smoke transport was based on the Reynolds Averaged Navier-Stokes, continuity, energy and species conservation equations. These governing equations in steady state may be written as:

$$\frac{\partial}{\partial x_j}\left(\rho U_i U_j\right) - \frac{\partial}{\partial x_j}\left[\mu\left(\frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i}\right)\right] = -\frac{\partial p}{\partial x_i} - \frac{\partial}{\partial x_j}\left(\rho\overline{u_i u_j}\right) + \rho\beta\left(T - T_{ref}\right) \tag{1}$$

$$\frac{\partial U_j}{\partial x_j} = 0 \tag{2}$$

$$\rho\frac{\partial}{\partial x_j}\left(U_i h\right) = \frac{\partial}{\partial x_j}\left(k\frac{\partial T}{\partial x_j} - \rho u_i h\right) + \frac{\partial U_i}{\partial x_j}\left(\mu\left(\frac{\partial U_i}{\partial x_j}\right) - \rho\overline{u_i u_j}\right) \tag{3}$$

$$\rho\frac{\partial}{\partial x_j}\left(U_i C\right) = \frac{\partial}{\partial x_j}\left(D_C\frac{\partial C}{\partial x_j} - \rho u_i C\right) \tag{4}$$

The numerical solution of the system of equations was accompanied by the Shear Stress Transport (SST) turbulence model to compute the Reynolds stress tensor, with a production limiter according to the formulation of Kato and Launder (1993) to avoid overproduction of turbulent kinetic energy in the impinging jet region. The turbulence model selection was the result of a validation process performed by Vittori et al. (2011).

The gravitational body force was included in equation 1 and buoyancy forces due to temperature variations were taken into account through the Boussinesq approximation. Since smoke is rather homogenous and miscible in air, it was considered to behave like gaseous species capable to spread in the air. Therefore, smoke concentration (C) was modeled by equation 4, with the CO diffusivity (D_{CO}) as recommended by Galdo-Vega et al. (2006). The governing equations were discretized using the element-based finite volume method. The domain was discretized through a finite element mesh grid used to construct the finite volumes employed to conserve mass and momentum. The model was run using ANSYS-CFX v.12. The simulations were executed on a PC with an Intel^{R} Core^{TM} 2 Duo E8400 3Ghz processor and 2Gb of RAM memory.

3. Multi-variable study

The domain configuration in Figure 1 allows to analyze the air curtain as an invisible barrier designed to ensure a target temperature and concentration difference between cells in the steady state. The present study was aimed to find how the air curtain may offer the best resistance to prevent smoke from escaping, and "confine" the hot air within the confinement cell. For this, the source mass flow rate was increased by 3 times, avoiding that extraction duct to take all the smoke; in this way some hot gases would remain in the cell trying to escape through the curtain. Finally, a Response Surface methodology was applied to analyze the influence of several parameters on the air curtain performance, based on a Design of Experiments approach, as a preliminary step prior to numerical optimization.

3.1 Design of experiments

The average discharge velocity (Vc), discharge width (Dc) and the cell length (Lc), were the independent parameters considered in the multi-variable study. The air curtain and extraction mass flow rates were set equal (Q_C=Q_E). This implies that the discharge velocity and extraction velocity are the same, meanwhile the curtain width represents the double length of the extraction duct, due to the tunnel symmetry. Figure 2a shows the combination of the independent variables based on a Central Composite Design and a G-optimality distribution (Box and Draper, 1987). The velocity range is in a plane normal to the figure, and the maximum (10 m/s) was based on the recommendation by Kashef et al. (2003).

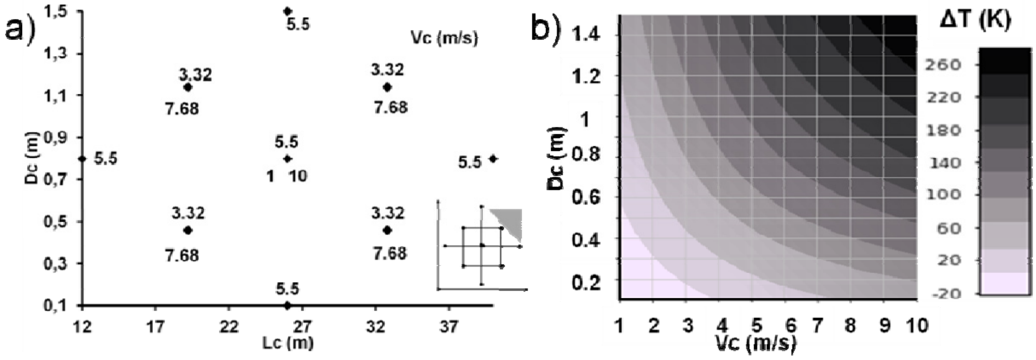

Figure 2: a) Design points for simulation. b) Response Surface for Lc=30 m.

3.2 Response Surface Model (RSM)

The output variables ΔT and ΔC are the absolute values of the difference between the average temperature and smoke concentration in the exit cell and the corresponding values in the confinement cell, respectively. Figure 2b shows the Response Surface calculated by the interpolation of a 2^{nd} order polynomial function using the "experimental" points shown in Figure 2a. The Response Surfaces for ΔT and ΔC have the same shape, which means that both output variables will respond the same way to variations of the entry variables, as result of the simulations. Figure 2b shows that for a larger curtain velocity and width, the temperature and concentration differences between cells are higher, indicating better performance of the air curtain. Since cell length (Lc) has little effect on the output variables, the results in Figure 2b are for a constant Lc.

3.3 Numerical optimization

The Response Surface allows studying the behaviour of the air curtain with respect to the entry variables through indexes or correlations between input and output variables. Since the objective is to determine the parameter combination that produces the best air barrier, in this case a simple inspection of the Response Surface reveals a global optimum for Vc = 10 m/s, Dc = 1.5 m and Lc = 30 m.

4. Analysis of results

Figure 3a shows the attenuation of temperature and smoke concentration achieved by the air curtain for the higher air curtain velocities and larger widths. It is possible to observe that for smaller widths, the solutions obtained by the RSM are very close to those simulated, revealing the advantage of the method where only 15 CFD simulations could produce the tendency of a system over a wide range of the independent variables. However, the solution for the curve of H/Dc=2 of the Response Surface diverges, because in this region the Response Surface was obtained by extrapolation of the "experimental" point, showing a weakness of the method. The verification with a simulation is required in those points to guarantee reliable solutions in the Response Surface. Figure 2a reveals a grey triangle outside of the design points, where the curve H/Dc_RSM=2 is located. Results indicate that curtain width is the more important factor because, for the same given velocity, the percentage of attenuation changes significantly from one curve to another. On the other hand, keeping constant the curtain width and changing the velocity, leads to smaller variations of the results.

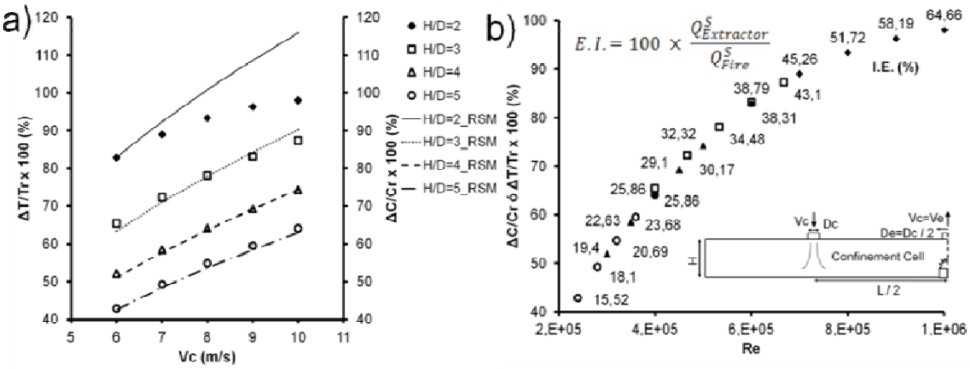

Figure 3: a) attenuation of smoke and heat transfer by the air curtain using the RSM approach and CFD calculations. b) Results as a function of Reynolds number (L/H=20).

Figure 3b reveals the air curtain behaviour through non-dimensional parameters. Results show how as the Reynolds number increases, the attenuation or the difference between cell conditions is larger, which means that the curtain acts as a better barrier. The attenuation of air temperature and smoke concentration are proportional as shown in Figure 3a. Figure 3b also shows the Extraction Index (E.I.) which represents the mass flow of smoke extracted by the ventilation with respect to the smoke generated by the source. The observed tendency regarding extraction is conformed to expectations when it is imposed Q_C=Q_E

The two points, near 60 % of attenuation, in Figure 3b reveal that the most important parameter is the air curtain mass flow rate which depends on the air curtain nozzle width and discharge velocity. The E.I. allows viewing the air curtain as a part of a ventilation system, where extraction plays an important role in the deviation of the air curtain flow. The extraction system minimizes jet deflections and takes out the smoke from the tunnel, as mentioned by Gupta et al. (2007).

Figures 4a and 4b show temperature contours in the tunnel for the best and the not-so-good curtains, respectively. The confinement cell exhibits a mean temperature of 540 K, that corresponds to the source temperature for both cases. The discharge width is the responsible of the curtain resistances, against the smoke flow because both have the same velocity (10 m/s). The air curtain in Figure 4b is totally deflected by the hot gases, whilst the barrier shown in Figure 4a depicts a stronger attachment to the tunnel floor, producing a lesser leakage through the lateral wall-jet after impingement.

The characteristic that reduces the curtain deviation is the potential core of the jet, which is only dependent of the jet nozzle width according to Cooper et al. (1992). It is defined by a constant speed (Vc), cone shape and length of h=6*Dc, from the discharge. Obviously, for H/Dc lower than 6, the curtain offer better confinement, because it has a bigger width that makes resistance to deviation and the potential core avoid mixing between the smoke and the air from curtain.

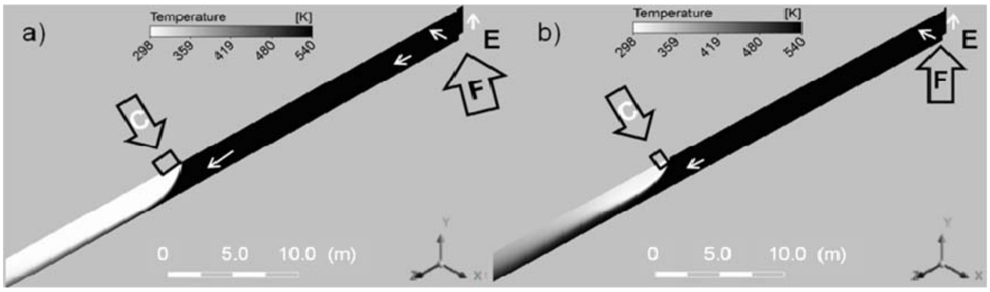

Figure 4: Temperature contour for Vc=10 m/s, a) H/Dc=2 and b) H/Dc=5. Where "C" is the curtain position, "E" the extractor and "F" the fire source location.

5. Conclusions

A numerical study of cellular confinement during a fire in tunnels by using air curtains was presented. A single cell confinement arrangement was tested by varying the relevant jet discharge velocity and width, observing its resistance to be crossed by smoke and heat, via CFD simulations, in steady state conditions. The Response Surface method was employed to study the behaviour of such a ventilation system by considering several variables over a rather wide range of values, reducing the number of computer simulations and thus, time consumption. This methodology was found helpful to investigate the air curtain behaviour, and focus on the role of the parameters that produce a more effective air barrier.

For the proposed 2-D domain, the length of the confinement cell has no influence on confining the smoke and heat propagation. The most effective air barriers are those operated at higher discharge Reynolds numbers. However, the most important parameter turns out to be the curtain width. The jet potential core is related to this parameter. A wider jet nozzle leads to a longer potential core and a more efficient air curtain even in case of jet deflection. Curtains or plane jets at lower Reynolds Numbers do not form efficient barriers because they do not hold significant temperature and smoke difference between contiguous cells, as revealed by the Response Surface.

The results presented in this paper provide useful information to design ventilation systems based on the use of air curtains. They allow select appropriate air curtain parameters and predict the attenuation capacity of the relevant systems. However, remain to study additional parameters as tunnel width to offer a final design proceed of air curtain as emergency ventilation system.

References

Box G., Draper N., 1987, Empirical model-building and response surfaces, John Wiley & Sons, First edition, USA, ISBN 0471810339.

Cooper D., Jackson D.C., Launder B.E., Liao G.X., 1992, Impinging jet studies for turbulence model assessment, Part I: Flow-field experiments, Int. J. Heat Mass Transfer, 36, 2675-2684.

Elicer-Cortés J.C., Demarco R., Valencia A., Pavageau M., 2009, Heat confinement in tunnels between two double-stream twin-jet air curtains, International Communications in Heat and Mass Transfer, 36(5), 438-444.

Galdo-Vega M., Ballesteros-Tajadura R., Santolaria-Morros M., 2006, Numerical 3D simulation of a longitudinal ventilation system: Memorial tunnel case, 2006 ASME Joint U.S. European Fluids Engineering Summer Meeting, July 17-20, Miami, Florida, USA.

Gobeau N., Ledin H.S., Lea C.J., 2002, Guidance for HSE Inspector: Smoke movement in complex enclosed spaces - Assessment of Computational Fluid Dynamics. <www.hse.gov.uk/research/hsl_pdf/2002/hsl02-29.pdf> Accessed 08.11.2011

Gupta S., Pavageau M., Elicer-Cortés J.C., 2007, Cellular confinement of tunnel sections between two air curtains, Building and Environment, 42, 3352-3365.

Kashef A., Bènichou N., Lougheed G., 2003, Numerical Modeling of Movement and Behavior of Smoke Produced from Fires in the Ville-Marie and L.-H.-La Fontaine Tunnels: Literature Review, Institute for Research in Construction. <www.nrc-cnrc.gc.ca/obj/irc/doc/pubs> Accessed 08.11.2011

Kato M., Launder B.E., 1993, The modeling of turbulent flow around stationary and vibrating square cylinders, Ninth Symposium on “Turbulent Shear Flows,” 10.4.1-10.4.6, Kyoto, Japan.

Mégret O., Vauquelin O., 2000, A model to evaluate fire characteristics, Fire Safety Journal, 34, 393-401.

Vittori F., Rojas-Solórzano L., Pavageau M., 2011, Comparative study of RANS turbulence models for impinging jet simulations. 9TH Engineer Annual Congress ASME USB 2011, 23 – 25 June, Caracas, Venezuela.