COMPUTATIONAL SIMULATION ON THE PERFORMANCE OF AIR PLANE JETS FOR SMOKE CONTROL

João C. Viegas*
Researcher
LNEC - Lisboa

Hildebrando Cruz
Research Fellow
LNEC - Lisboa

ABSTRACT

This communication analyses the possibility of use of air curtains approximately vertical to prevent the smoke flow to outside of fire compartments through vertical permanent slots (doors). Computational simulations have been developed, using the program Fire Dynamics Simulator, where several conditions relevant to smoke-tightness have been adopted. It was concluded that it is possible to obtain smoke-tightness through the adjustment of the parameters of the plane jet, in association with smoke exhaust from the compartment.

KEY-WORDS: Smoke control; Plane jets; Air curtains; CFD; FDS.

1. INTRODUCTION

The smoke flow inside buildings is a major cause of death in the event of fires. The technology currently used to prevent this event relies on the enclosure of building spaces by fire resistant walls, on the use of fire resistant doors and the use of smoke control. In many cases closing passageways with fire doors makes difficult to identify the escape route and can delay people egress; in other cases the use of fire gates makes difficult to use them as escape routes When the spread of fire through the void is unlikely, it is acceptable the use of an air curtain if it is effective in stopping the smoke flow. Air curtains do not impair the visibility of building occupants in evacuation and do not cause difficulties for people using escape routes.

*Corresponding author – Dep. de Edifícios, Laboratório Nacional de Engenharia Civil. A. do Brasil, 101, 1700-066 Lisboa.
Telef.: +351 218 443 533, Fax: +351 218 443 024, E-mail: jviegas@lnec.pt

There are several applications of this concept in tunnels [1] and in building corridors [2], but these are based on the pull-push principle applied to horizontal air curtains. Several other authors studied the application of single vertical air curtains (upwards or downwards) [3, 4, 5, 6, 7, 8] and there is also research on double vertical air curtains [9]. Some studies used CFD simulations to assess the performance of air curtains relating to curtain tightness on corridors [8], to fire/explosion accident in a clean room [10] and to contaminants dispersion from clean rooms [11]. In Portugal a study was carried out to evaluate the performance of air curtains to separation of different environmental conditions [12], however it reports temperature differences much lower than the expected in case of fire.

The mentioned research is not presenting clearly the need of smoke exhaust in the fire compartment, which the authors of this paper consider to be a key issue for the success for the smoke tightness by air curtains for high temperature smoke. Therefore, it is clear the need of research under this topic in order to apply air curtains in a more general way to the open boundaries of fire compartments.

This project has the objective of developing and applying the technology of air curtains to limit the smoke flow through the building openings. The methodology followed in this research includes: (i) the development of an analytical model that relates the relevant characteristic quantities of a plane jet with the characteristics of the environment in which the fire develops, (ii) small scale experiments with salt water modelling to assess the parameters that control the smoke tightness of the air curtain, (iii) CFD simulations to assess the performance of a full scale air curtain near a fire source and (iv) fire experiments with a full-scale test specimen. In this paper the analytical model and saltwater experiments are presented.

The analytical model assumes that the pressure due to the jet momentum balances the pressure due to buoyancy at the opening. This model also includes a simplified analytical methodology for estimating the temperature inside the fire compartment. It was already presented in a previous paper [13].

In that previous work several vertical downward plane jets and horizontal plane jets were tested in a small-scale model (1/20). The buoyancy due to fire was reproduced by density difference between saltwater and freshwater. The model, made by Plexiglas, includes one compartment connected to outside by a single opening which is protected by the plane jet. The similarity laws allow extrapolate that results to real fire cases. In these tests, the continuous variation of test parameters (exhaust flow rate and jet velocity) allowed to find the limiting values compatible with the smoke tightness for the set of parameters adopted during the test (water density, jet angle, jet thickness).

The results showed that it is feasible to avoid the leakage of the denser fluid (simulating smoke) to outside of the compartment using a plane jet (curtain) flowing a less dense fluid. The tests clearly showed that the plane jet is critical to the tightness because when it is stopped (with no change of other relevant parameters, including exhaust flow rate and the intensity of the saltwater source) the loss of tightness occurs immediately. Thus, the test results demonstrate the feasibility of the use of air curtains to control smoke and allow the experimental assessment of the parameters that control tightness.

In this paper CFD simulations corresponding to the full scale case are presented. For every simulation it is not feasible to perform the continuous variation of parameters (as it was done in small scale testing). It is just possible to change parameters from simulation to simulation until smoke tightness is obtained (by trial and failure procedure). Therefore, it is not possible to find the limiting values of the parameters, but just an interval where the limiting values of the parameters are lying. This paper present that work and confirm that is feasible to use plane jets of clean air to avoid the smoke spread.

2. METHODS

2.1 Analytical model

The analytical model was previously developed and presented elsewhere [13]. Hereafter just the main steps will be presented.

The smoke tightness due to an air curtain (plane jet) is based on the balance of the air curtain momentum and the momentum of the smoke flow. The nozzle of the plane jet is put at the door soffit level and is flowing downward, being $\propto_0$ the angle between the curtain axis and the vertical plane. This work considers that the jet momentum is conserved. Since the smoke flow (this case is horizontal) is normal to the plane of the opening (vertical), only the momentum due to the horizontal component of jet velocity is concerned. Thus, the smoke tightness is reached if the value of the pressure difference ΔP_s due to the difference in fluid density between indoor and outdoor (assuming uniform density in each environment) is balanced by the pressure difference ΔP_a developed by momentum ($\Delta P_a/\Delta P_s \geq 1$). The ratio $\Delta P_a/\Delta P_s$ is given by equation (1).

$$\frac{\Delta P_a}{\Delta P_s} = D_m \mathrm{sen} \propto_0 \tag{1}$$

where D_m is the deflection modulus, which is defined by the equation (2).

$$D_m = \frac{\rho_0 b_0 u_0^2}{g h^2 (\rho_0 - \rho_1)} \tag{2}$$

where ρ_0 is the outdoor density, ρ_1 is the indoor density, b_0 is the thickness of the jet nozzle, u_0 is the initial jet velocity, g is the gravity acceleration and h the height above the neutral plane (when the difference in densities is uniform, the pressure difference varies linearly with the height).

The smoke exhaust from the compartment generates an inlet velocity at the door u_a , which momentum shall be also considered. Therefore, the equation (1) takes the form of the equation (3).

$$\frac{\Delta P_a}{\Delta P_s} = \frac{\rho_0 b_0 u_0^2 \mathrm{sen} \propto_0 + \rho_0 h u_a^2}{g h^2 (\rho_0 - \rho_1)} \tag{3}$$

Saltwater modelling showed that

$$u_a \geq Au_0 \, b_0^{0.5} \tag{4}$$

$$\frac{\Delta P_a}{\Delta P_s} = \frac{\rho_0 b_0 u_0^2}{gH^2(\rho_0 - \rho_1 *)} \left(\text{sen} \propto_0 + HA^2\right) \geq 0.0184 \tag{5}$$

$$u_0 = \sqrt{\frac{0.0184 \, gH^2(\rho_0 - \rho_1 *)}{\rho_0 b_0(\text{sen}\propto_0 + HA^2)}} \tag{6}$$

where A is a constant, H is the full height of the door and $\rho_1 *$ is the density of the smoke source (saltwater). In saltwater modelling it was not possible to assess directly the indoor density ρ_1 and the smoke layer depth; therefore, the threshold value $\Delta P_a/\Delta P_s = 0.0184$ is much lower than the expected. However, saltwater modelling confirmed the general form of the equations to be used. CFD and full scale experiments have to be used to assess this threshold value.

In a fire, the minimum exhaust flow rate $\dot{V}_{exaust}$ shall include the thermal expansion and a portion proportional to the jet flow rate (4), according to the equation (7):

$$\dot{V}_{exaust} = \frac{\dot{Q}_c}{\rho_0 \overline{C_p} T_0} + Cb_0{}^{0,5} u_0 \tag{7}$$

being C a constant of proportionality (that includes the geometry of the opening) to be assessed by experiments, $\dot{Q}_c$ is the convective part of the heat release rate, $\overline{C_p}$ is the average specific heat at constant pressure (considering here the average is an approach, which does not have significant consequences since the final equations will be adjusted by empirical coefficients) and T_0 is the initial temperature. The absolute temperature of the hot fluid (T_1) is then calculated according to the equation (10). The final temperature dependence on C_{p1} requires an iterative solution.

$$T_1 = \frac{C_{p0} T_0}{C_{p1} - \frac{\dot{Q}_c}{\rho_0 T_0 \dot{V}_{exaust}}} \tag{10}$$

There are a number of methods that allow estimating the position of the neutral plane (eg, see [14]). The results of the experiments show that the flow near the jet is quite complex and that the neutral plane may be strongly disturbed.

2.2 CFD characteristics

The computer code Fire Dynamics Simulator (FDS) was used for CFD modelling [15]. FDS solves numerically the equations of the fluid mechanics and thermodynamics. It is adequate for Low Mach Number applications and it has the following characteristics:

- Uses the large-eddy simulation (LES) turbulence model;
- Uses an explicit, second-order, kinetic-energy-conserving numeric scheme;
- Uses a structured, uniform, staggered grid;
- Adopts simple immersed boundary method for treatment of flow obstructions;
- Adopts constant turbulent Schmidt and Prandtl numbers;
- Uses eddy dissipation concept (fast chemistry) for single-step reaction between fuel and oxidizer;
- Adopts grey gas radiation model with finite volume solution to the radiation transport equation.

The fire compartment is 5.20 m long (X), 4.60 m large (Y) and 2.40 m high (Z). There is a door 2.00 m high and 0.90 m wide located in the wall Y = 0 m and starting at X = 4.00 m. The extraction slot is 0.30 m high and 1.00 m wide and is located in the wall Y = 4.60 m and starting at X = 2.00 m and Z = 2.00 m. The heat source is located on the floor between X = 1.00 m, X = 2.00 m, Y = 2.00 m and Y = 2.70 m. The total Heat Release Rate (HRR) is 700 kW. The calculation domain is subdivided into 5 subdomains, according to table 1.

Table 1: Grid characteristics

Mesh	Cell number			Coordinates [m]					
number	I	J	K	Xinitial	Xfinal	Yinitial	Yfinal	Zinitial	Zfinal
1	52	4	24	0.00	5.20	-0.80	-0.40	0.00	2.40
2	38	12	24	0.00	3.80	-0.40	0.80	0.00	2.40
3	28	12	48	3.80	5.20	0.20	0.80	0.00	2.40
4	56	94	96	3.80	5.20	-0.40	0.20	0.00	2.40
5	52	38	24	0.00	5.20	0.80	4.60	0.00	2.40

The finest grid is used at the door; the smallest cell is 0.025 m x 0.006 m x 0.025 m; the total number of cells is approximately 585000. The unsteady simulations were running until steady state was reached (aprox. 200 s). Table 2 presents the characteristics of the simulations carried out. The jet pitch angle is the angle between the central plane of the jet and the vertical plane of the door.

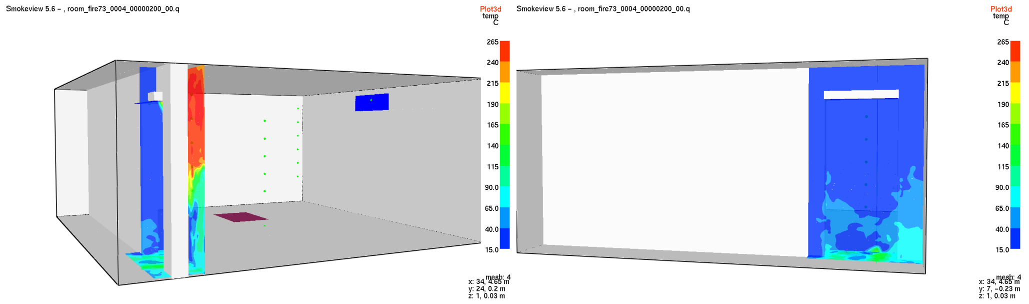

Figure 1: Simulation 73. Temperature field near the plane jet. Perspective (left) and front view of the door plane (right)

Table 2: Simulations characteristics

Simulation number	HRR [kW]	Jet thickness [m]	Jet initial velocity [m/s]	Jet pitch angle [º]	Exhaust flow rate [m^3/s]	Comments
86	0	0.025	17.50	0	0	Validation
87	0	0.025	17.50	30	0	Validation
73	700	0.100	11.50	30	1.869	Not smoke tight
74	700	0.100	7.03	30	2.391	Not smoke tight
75	700	0.100	7.03	30	3.261	smoke tight
79	700	0.025	17.50	20	3.228	smoke tight
80	700	0.050	11.15	25	3.228	smoke tight

2.3 CFD validation

FDS is a computer code made available to the scientific community by the National Institute of Standards and Technology. Several validation cases show that this computer code is adequate for the simulation of the fire development in buildings [16].

Two simulations of a plane jet were carried out (simulations 86 and 87 in Table 2). The plane jet was located at door and has the same characteristics of the air curtain to retain the smoke; however in these simulations the heat source was not activated and the smoke exhaust was also inactive. In the first simulation the centre plane of the jet is vertical; therefore it is aligned with the mesh. In the other simulation the centre plane of the jet has a pitch angle of 30º with the vertical plane. This is the maximum pitch angle adopted in further simulations; therefore, it is expected that the performance for pitch angle within this range is falling between the results of these two simulations.

The simulation results were compared with the analytical formula for free plane jet (11) [17]:

$$\mathrm{W} = 3.4\left(\frac{b_0}{2x}\right)^{0,5} W_0 e^{-57(y/z)^2} \quad (11)$$

where W is the jet longitudinal velocity, b_0 is the width of jet nozzle and W_0 is the jet velocity at the nozzle. While the analytical formula represents the average longitudinal velocity field, the simulation results correspond to instantaneous velocity field; therefore, some differences were expected due to the turbulent behaviour of the jet. Moreover, the simulated jets may be affected by the compartment geometry (proximity of walls, presence of the floor, etc.), thus it is not expected that the results of the simulations are directly comparable with the free jet formula, but this comparison contributes to the assessment of the goodness of the simulation results.

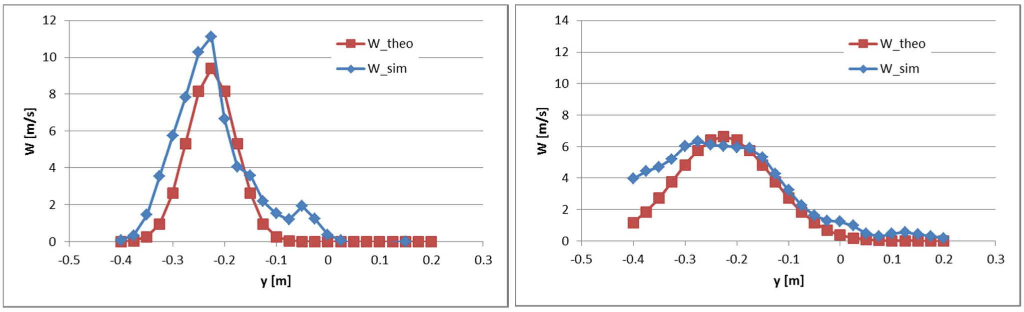

Figure 2: Simulation 86. Comparison of predicted longitudinal jet velocity with theory. 0.5 m from nozzle (left) and 1.0 m from nozzle (right)

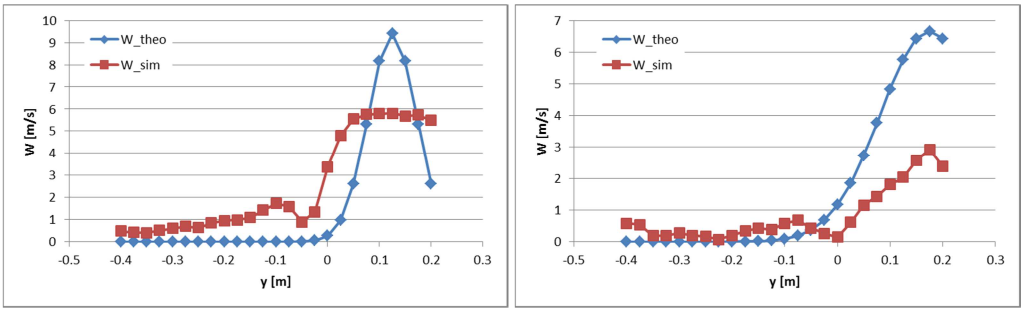

Figure 3: Simulation 87. Comparison of predicted longitudinal jet velocity with theory. 0,5 m from nozzle (left) and 1,0 m from nozzle (right)

The comparison between predicted longitudinal jet velocity with theory (11) at jet pitch angle of 0º (figure 2) show that simulation is less dissipative than theory, but the results are close to theory. This shows that the boundaries of the compartment do not have a strong effect in these results. However, when jet pitch angle is 30º the predictions are much more dissipative. As the jet pitch angle of simulations are laying between 20º and 30º it is expected that the predicted velocities are lower than the real velocities. In this case, it is expected that would be possible to get the smoke tightness for jet nozzle velocities lower than the adopted in the simulations.

3. Analysis of the results

3.1 Smoke tightness

In the last column of Table 2 are listed the results of smoke tightness at the door. For simulations 73 and 74 there is smoke flow through the door, despite the air curtain jet being active. Figure 1 presents, as an example, the temperature field at the door plane of simulation 73 for the steady state. The smoke flows to outside near the floor, were the jet velocity is lower. The smoke tight condition was reached for simulations 75, 79 and 80. Figure 4 presents, as an example, the temperature field at the door plane of simulation 75 for the steady state. It is clear that the door is smoke tight. Table 3 present the characteristics assessed for every simulation, where $\overline{T}_{smoke}$ and $\bar{\rho}_{smoke}$ are the average temperature of density of the smoke, h is the smoke depth below the door soffit, $\dot{V}_{door}$ is the flow rate through the door, u_a is the inlet velocity at the door and $\dot{V}_{exaust}$ is the exhaust flow rate. It is also shown the difference between exhaust flow rate and the flow rate through the door, which represent the thermal expansion. It is interesting to note that the thermal expansion is not equal for every simulation and is depending of the ventilation conditions.

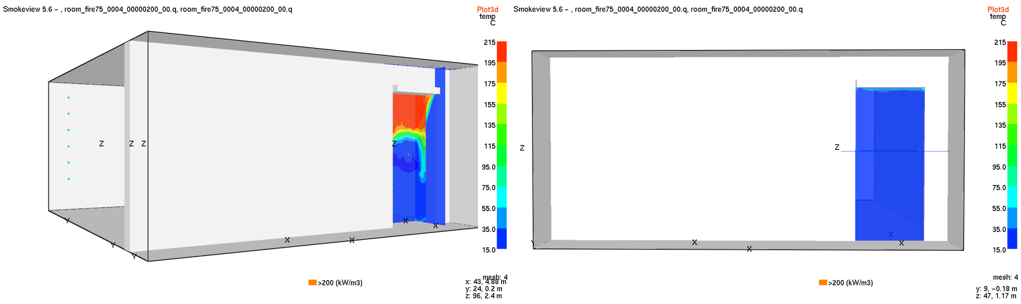

Figure 4: Simulation 75. Temperature field near the plane jet. Perspective (left) and front view of the door plane (right)

Table 3: Simulations results

Simulation number	$\overline{T}_{smoke}$ [ºC]	$\bar{\rho}_{smoke}$ [kg/m³]	h [m]	$\dot{V}_{door}$ [m^3/s]	u_a [m/s]	$\dot{V}_{exaust}$ [m^3/s]	$\dot{V}_{exaust} - \dot{V}_{door}$ [m^3/s]	Comments
73	243	0.670	1.05	0.951	0.528	1.869	0.918	Not smoke tight
74	198	0.734	1.45	1.107	0.615	2.391	1.284	Not smoke tight
75	203	0.726	0.55	2.282	1.268	3.261	0.979	smoke tight
79	197	0.735	0.55	1.845	1.025	3.228	1.383	smoke tight
80	179	0.765	0.55	2.332	1.295	3.228	0.896	smoke tight

3.2 Smoke temperature and smoke depth

The smoke temperature varies inside the compartment; therefore a criterion to assess the average temperature is needed. The vertical curves of the temperature in the symmetry plane of the door were analysed for Y = 0.2 m and Y = 0.8 m (figure 5). The hot layer limit (interface layer between smoke and lower cold layer) is clearly seen for temperature curves at Y = 0.2 m, but it is not so clear for the curves at Y = 0.8 m. The curves at Y = 0.2 m show, bellow the interface layer some high temperature peaks, that correspond to the development of vortices. From simulations it is clear that such vortices tend to dominate the flow below the smoke layer to the floor and may cause the loss of smoke tightness. The estimated smoke depth bellow the door soffit is indicated in Table 3. The corresponding average temperature was estimated for this smoke thickness but for Y = 0.8 m, where the influence of the jet turbulence on the smoke layer is lower. These results are also shown in table 3.

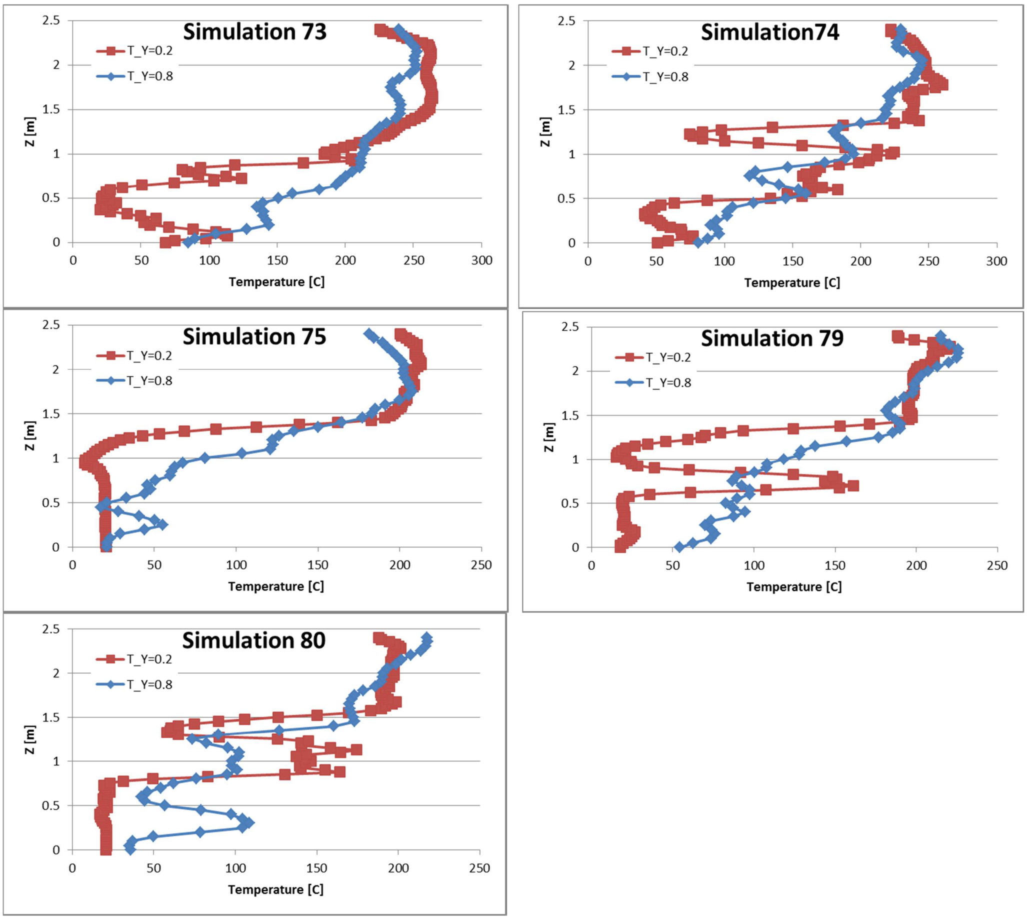

Figure 5: Temperatures at symmetry plane of the door for Y = 0.2 m and Y = 0.8 m

3.3 Pressure forces and exhaust flow rate

Table 4 shows the assessment of , according to equation (3), using the simulation results. For simulations where smoke tightness was reached . For simulations where smoke tightness was not reached . These values are much higher than the value obtained for saltwater modelling (see equation (5)), but in that case the saltwater source conditions were used instead of the smoke layer conditions; therefore, the threshold value obtained from simulations shall be closer to the real value. We stress that due to dissipative behaviour observed in simulations, it is expected that real values are lower. These values allows the calculation of , according to equation (3).

The flow rate $\dot{V}_{jet}$ of the plane jet is given by equation (12) [17]:

$$\dot{V}_{jet} = 0{,}44\left(\frac{2x}{b_0}\right)^{0{,}5}\dot{V}_0 = 0{,}44\left(\frac{2x}{b_0}\right)^{0{,}5} 0.5\, u_0\, b_0 w = 0{,}44(0.5\, b_0 x)^{0{,}5} u_0\, w \qquad (12)$$

where x is the distance from the nozzle, b_0 is the thickness of the nozzle, w is the width of the nozzle (and door) and $\dot{V}_0$ is the jet flow rate at the nozzle. The corresponding average velocity at the door u_{a_min} is given by equation (13), considering that the full length of the jet is limited by the floor, where H is the full height of the door. In Table 4 it is shown the ratio u_a/u_{a_min}. For simulations where smoke tightness was reached $u_a/u_{a\_min} \geq 1.63$. For simulations where smoke tightness was not reached $u_a/u_{a\_min} \leq 1.17$. The constant C of equation (7) may be assessed from these conditions.

$$u_{a\_min} = \frac{0{,}44}{(H\cos\propto)^{0.5}}(0.5\, b_0)^{0{,}5} u_0 \qquad (13)$$

The thermal expansion $\dot{V}_{exaust} - \dot{V}_{door} = \dot{Q}_c/\rho_0\overline{C_p}T_0$ was calculated and is shown in Table 3. Using this value, the ratio between total heat release rate and convective heat release rate may be assessed, according to equation (14). The higher value is 0.70, that agrees with common values accepted for higher convective fraction of heat release rate.

$$\dot{Q}_c/\dot{Q} = \frac{(\dot{V}_{exaust} - \dot{V}_{door})\rho_0\overline{C_p}T_0}{\dot{Q}} \qquad (14)$$

These values allow the estimation of the range where the parameters relevant for the calculation of an air curtain for smoke retention (namely u_0 , u_a and $\dot{V}_{exaust}$) shall lay.

Table 4: Simulation analysis

Simulation number	$\Delta P_a/\Delta P_s$	u_{a_min} [m/s]	u_a/u_{a_min}	$\dot{Q}_c/\dot{Q}$	Comments
73	1.45	0.86	0.61	0.46	Not smoke tight
74	0.38	0.53	1.17	0.65	Not smoke tight
75	2.86	0.53	2.41	0.49	smoke tight
79	2.78	0.63	1.63	0.70	smoke tight
80	3.30	0.58	2.25	0.45	smoke tight

4. CONCLUSIONS

The results obtained show that it is feasible to restrict the flow of smoke to inside of the enclosure using a plane jet of cold air at the opening (air curtain). The simulations showed that the existence of the plane jet is critical to smoke-tightness of the opening because when it is too weak the smoke tightness fails. Thus, the results of the simulations demonstrate the feasibility of the use of air curtains for smoke control.

ACKNOWLEDGMENTS

This research was undertaken as part of a research project supported by the ADI – Agência de Inovação under Grant QREN n.º 23226 (Smoke Shield).

REFERENCES

[1] Gupta S., Pavageau M., Elicer-Cortés J.C. - *Cellular confinement of tunnel sections between two air curtains*, Building and Environment 42, 2007, p. 3352–3365.
[2] Sakurai H., Hayashi T., Shibata, Kanehara K. - *Researches on air shutter for fire defence*, Fire Safety Journal 2, 1979/80, p. M. 9–16.
[3] Cumo F., Rossetti S., Guidi G. - *3D simulation of dynamic barriers against fume and gaseous toxic substances*, Proceedings of the Ventilation 2000 - 6th International Symposium on Ventilation for Contaminant Control, 4-7 June 2000, Helsinki, Finland.
[4] Dufresne de Virel M., Solliec C., Guyonnaud L. - *Mise en sécurité contre fumées d'incendie*, Revue Face au Risque 348, 1998, p. 13–18.
[5] Altinakar M.S., Weatherill A. - *Use of an inclined air curtain for preventing smoke propagation in a tunnel during fire emergency*, Proceedings of the 4th International Conference on Safety in Road and Rail Tunnels, 2001.
[6] Gugliermetti F., Santarpia L., Zori G. - *Air curtain applied to fire smoke pollution control*, air pollution 2003, 11th International Conference on Modelling, Monitoring and Management of Air Pollution, September 17–19, 2003, p. 541–549, Catania, Italy.
[7] Sawley M., Greveldinger B., Drotz A. - *Numerical flow simulation of an air curtain for road tunnel fire safety*, Supercomputing Rev 14, 2004, p.3–6.
[8] Hu L.H., Zhou J.W., Huo R., Peng W., Wang H.B. - *Confinement of fire-induced smoke and carbon monoxide transportation by air curtain in channels*, Journal of Hazardous Materials 156, 2008, p. 327–334.
[9] Felis F., Pavageau, Elicer-Cortés J.C., Dassonville T. - *Simultaneous measurements of M. temperature and velocity fluctuations in a double stream-twin jet air curtain for heat confinement in case of tunnel fire*. International Communications in Heat and Mass Transfer 37, 2010, p. 1191–1196.
[10] I Y.P., Chiu Y.L., Wu S.J. - *The simulation of air recirculation and fire/explosion phenomena within a semiconductor factory*. J Hazard Mater;163, 2009, p. 1040 - 51.
[11] Shih Y.C., Yang A.S., Lu C.W. - *Using air curtain to control pollutant spreading for emergency management in a cleanroom*. Building and Environment 46, 2011, p. 1104 – 1114.
[12] Neto L.P.C. - *Estudo de sistemas de vedação aerodinâmica do tipo cortina de ar*. Coimbra: Faculdade de Ciências e Tecnologia da Universidade de Coimbra, Departamento de Engenharia Mecânica. Dissertação para Doutoramento em Engenharia Mecânica (Aerodinâmica). 2006.

[13] Viegas J.C., Cruz H. - *Saltwater experiments of air curtains to smoke control in fires*. V Conferência Nacional de Mecânica dos Fluidos, Termodinâmica e Energia. MEFTE 2014, 11–12 Setembro 2014, Porto, Portugal.

[14] Viegas J.C. - *Utilização de ventilação de impulso em parques de estacionamento cobertos*. Lisboa: Laboratório Nacional de Engenharia Civil. Série Teses e Programas de Investigação, 55. 2008.

[15] McGrattan K. et al. - *Fire Dynamics Simulator. Technical Reference Guide. Volume 1: Mathematical Model*. National Institute of Standards and Technology. Gaithersburg, USA Special Publication 1018. 2013.

[16] McGrattan K. et al. - *Fire Dynamics Simulator. Technical Reference Guide. Volume 1: Validation*. National Institute of Standards and Technology. Gaithersburg, USA Special Publication 1018. 2014.

[17] Blevins, R.D. - *Applied fluid dynamics handbook*. Van Nostrand Reinhold Company Inc., New York, USA, 1984.