Proceedings of BS2013:
13th Conference of International Building Performance Simulation Association, Chambéry, France, August 26-28

EFFICIENCY OF AIR CURTAINS USED FOR SEPARATING SMOKE FREE ZONES IN CASE OF FIRE

Grzegorz Krajewski
Building Research Institute, Fire Research Department, Warsaw, Poland

ABSTRACT

The aim of thise paper is to take the advantage of CFD application in calculating, optimising, and designing air curtains used to separate smoke free zones in case of fire. Properly designed air curtain produces a pressure drop which forbids transversal flow through the opening. It is hard to make a good quality CFD calculation of that kind of air curtain because of a high velocity and relatively thin nozzle. Most air curtains are tested on scaled down models which are difficult to extrapolate. Author of that article had done tests in a real scale model. Tests results were used to verify chosen turbulence model. The intention of this paper is to present the possibilities of CFD while designing air curtains used in fire safety engineering.

INTRODUCTION

To avoid standstill and facilitate the flow of vehicles and people through doorways of buildings and other enclosures, solid doors are often replaced or supplemented by air curtains (air screens, air planes). Simultaneously, the air screen eliminates or reduces the transfer of heat and mass through the opening. Air curtains have become popular in the 60's of the 20th century; nevertheless, the principles of the air planes dates back to 1904. The concept of air screens was founded by Theophilus van Kennel and his idea has become a forerunner of modern air curtains. The flow of air across the doors is caused by: the difference of pressure between two volumes of fluid, the dissimilar temperature values, and the presence of ventilation system. Air curtain devices are often used in the entrances to the public buildings, cooling rooms and refrigerators, as well in chemical and electronic industry.

The knowledge that a direct exposure to fire is not the most immediate threat to people's lives, has been displayed by previous experience and research. A vast majority of fatalities connected with fire are triggered by the smoke-inhalation. Therefore, to decrease the number of fatalities air curtain devices can be used as virtual screens to stop smoke spreading in a building object. Safe evacuation of people and secure intervention of fire fighters are of significant importance and need to be taken into consideration, as mentioned in the article of Sztarbala and Krajewski (2009). Air screens can be used to obstruct or reduce the movement of toxic smoke while enabling full access to emergency exits. Properly designed air curtains produce a pressure drop which forbids transversal flow through the opening.

The main criterion for the air curtain is its efficiency which is the rate of mass and heat transfer while crossing it, in comparison with the same opening without the curtain.

Furthermore, air- curtains designing is considered relatively difficult. It is essential to know the conditions on both sides of the air curtain to chose its parameters correctly.

If the outlet velocity is too high and blowing angle is not optimal, the air curtain can increase the heat and mass transfer through itself. In other case, if the jet velocity is too low, air curtain will not be tight enough.

Currently, most of the installations are experimentally set up on scaled down or full scale physical models. Nonetheless, it is hard to extrapolate results from the scaled down model to the real geometrical dimensions because the Euler number similitude is unavailable. As it is written in published literature that kind of extrapolation provides an over-efficient results.

To study the performance of air curtains the application of computational fluid dynamics (CFD) is immensely useful. It is critical to properly define initial and boundary conditions and compare gained results from simulations with analytical equations.

AIR CURTAIN DESCRIPTION

Velocity distribution

There are numerous publications involving experimental data and mathematical analysis presenting the theory of a free stream jet as velocity profile and deflection of the centreline axis. Significantly in this subject, publications of Abramovich (1963) and Rajaratnam (1976) offer the most fundamental piece of information.

Particularly, depending on the height and the stream of an air, a jet shows two, three or four regions. It is possible to distinguish the potential core zone, the transition zone, and the developed zone or the impinging zone (Fig.1):

Potential core zone - characteristic for this region is that the centreline velocity is almost constant and equal to the outlet velocity U_0.

Transition zone – this region starts with the velocity decay and the amplification of the jet expansion. It generally starts after approximately 5 e from the nozzle. Analytical solution of the velocity can be described by:

$$\frac{U(x,y)}{U_0} = \frac{1}{2}\left[1 + erf\left(\sigma_1 \frac{y + \frac{e}{2}}{x}\right)\right] \qquad \text{Eqs. 1}$$

Developed zone – in this region velocity decay remains constant. Velocity decay expressed with non-dimensional quantities. It generally starts after approximately 20 e from the nozzle. Analytical solution of the velocity can be described by (Schlichting 1968):

$$\frac{U(x,y)}{U_0} = \frac{\sqrt{3}}{2}\sqrt{\frac{7.67e}{x}}\left[1 + \tanh^2\left(7.67\frac{y}{x}\right)\right] \qquad \text{Eqs. 2}$$

$$\frac{U_C(x)}{U_0} = C_1\left(\frac{x}{e} - C_2\right)^{-\frac{1}{2}} \qquad \text{Eqs. 3}$$

C_1 and C_2 depends on the nozzle shape and on the boundary conditions. They are in the range $1.9 < C_1 < 3.0$ and $-8 < C_2 < 10$

Impinging zone - this region is in the vicinity of the floor. The flow in that zone is very complex and still not well known. Thickness of that zone is approximately 15% of total height.

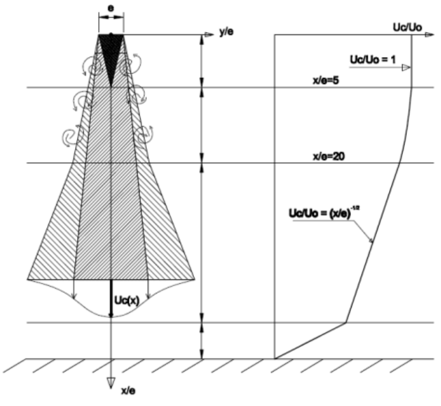


Figure 1 Zones of a free air jet

Deflection of jet axis

According to F. C. Hayes and W. F. Stoecker, the problem of a plane jet, subjected to a lateral side pressure and the blowing angle directed to the side of higher value of the pressure is treated in the literature with the conclusion that the axis of the jet represents a circular curvature.

The x and y momentum equations for the control volume are:

$$\Delta p dy = -(\rho_o / g_c) b_o u_o^2 \sin(\alpha) d\alpha \qquad \text{Egs. 4}$$

$$\Delta p dx = (\rho_o / g_c) b_o u_o^2 \cos(\alpha) d\alpha \qquad \text{Egs. 5}$$

These equations are true when it is assumed that only external forces are the pressure forces acting on a control volume (Fig.2).

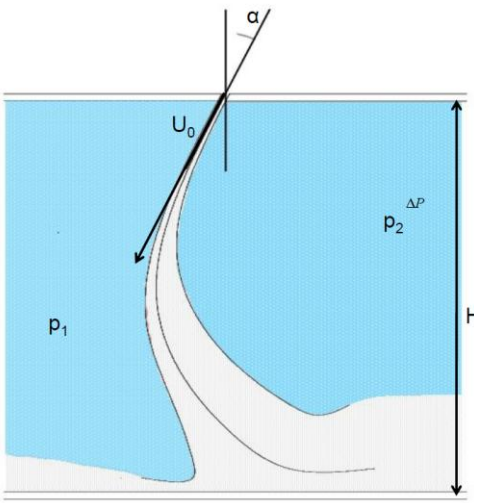


Figure 2. Air curtain

Integrating the first and the second equation we can find the coordinates of the jet centreline x_c and y_c. Finally, we receive the equation of the jet axis arc's shape. Already mentioned analytical equations were numerously confirmed by results of experimental research.

$$y_c = \frac{\rho_o b_o u_o^2}{g_c \Delta p}(\cos\alpha_o - \cos\alpha) \qquad \text{Eqs. 6}$$

$$x_c = \frac{\rho_o b_o u_o^2}{g_c \Delta p}(\sin\alpha - \sin\alpha_o) \qquad \text{Eqs. 7}$$

$$\left(\frac{g_c \Delta p}{\rho_o b_o u_o^2} x_c + \sin\alpha_o\right)^2 + \left(\frac{g_c \Delta p}{\rho_o b_o u_o^2} x_c - \cos\alpha_o\right)^2 = 1 \qquad \text{Eqs. 8}$$

EXPERIMENT

Before studying the air curtain, used to prevent smoke movement, the verification of numerical model has to be done. The verification was done in a tunnel with dimensions 8 m long, 1 m width and a height 2 m.

Tests equipment enable to carry tests with the blowing angle of the jet equalling 0 - 45° and velocity up to 30 m/s. Nozzle weight is 20 mm but it can be changable. The pressure difference on both sides of air curtain could be set in a range from 0 Pa to 200 Pa.

Schematic drawing of the test facility is presented below in Figure 3.

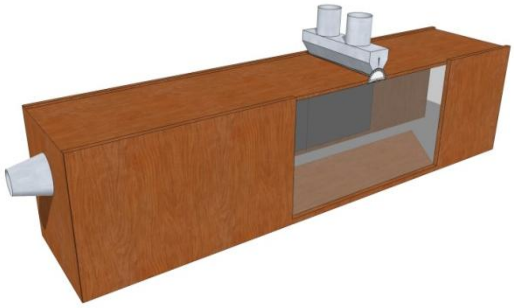


Figure 3 Schema of test facility made

Figure 4 Real test tunnel in laboratory

The free air jet

The initial test, which was done to validate numerical model, was a free jet test. Testes were carried out for three different air velocities at the nozzle outlet and equal 10 m/s, 20 m/s and 30 m/s. Referring to free jet centreline velocity decay and transversal distribution of the jet velocity were interpreted.

The velocity measurements in the centreline were done using hot wire thermo anemometry with three wires to measure velocity in two directions (Figure 5).

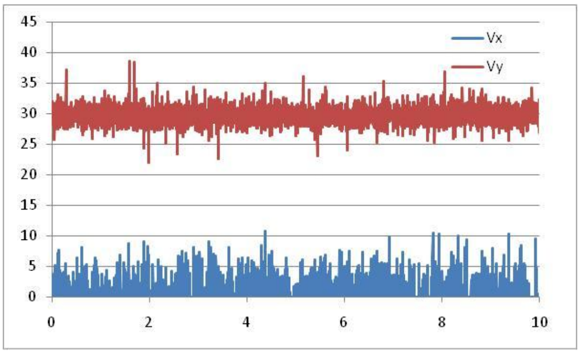


Figure 5 Velocity at the nozzle outlet

Additionally some visualisations of the stream were done to present its shape and turbulence (Figure 6).

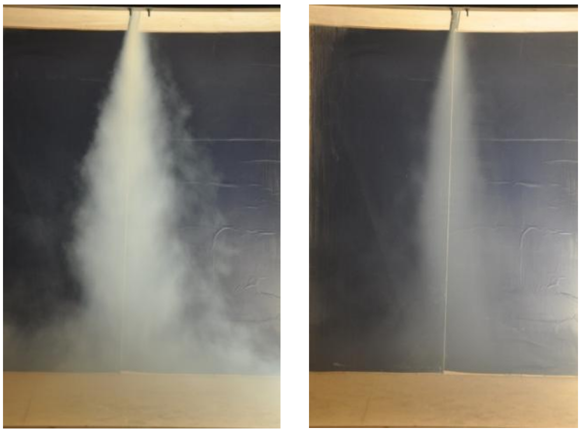


Figure 6 Air curtain visualisation

SIMULATION

In order to evaluate the parameters and the effectiveness of air curtain, it is necessary to perform series of calculations using the computational fluid dynamics application.

To confirm the correctness of boundary conditions and CFD models application, numerical calculations of free jet were performed. Defined initial conditions were the same as in the experiment. Blowing angle of the jet was equal 0°,outlet velocity from the nozzle 10 m/s, 20 m/s and 30 m/s. Turbulence intensity was the same as in experiment and was equal about 5%.

The three-dimensional model of the analyzed domain was build as it was in the experiment. In the middle of the ceiling an air curtain outlet was created. The domain has been divided into finite number of control volumes using an unstructured hexahedral grid. The total quantity of control volumes was approximately 2 500 000 with dimensions ranging from 2 mm in the area of air curtain outlet to 20 mm on the peripheries of the domain. Three-dimensional model is presented on Figure 7.

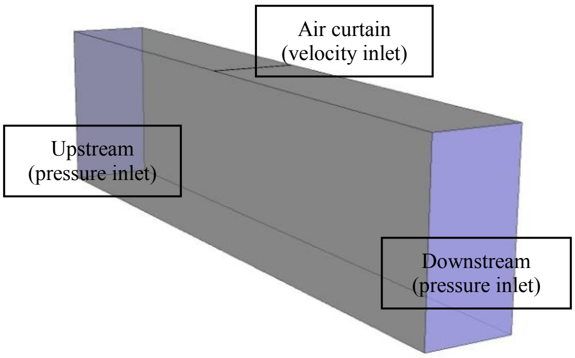
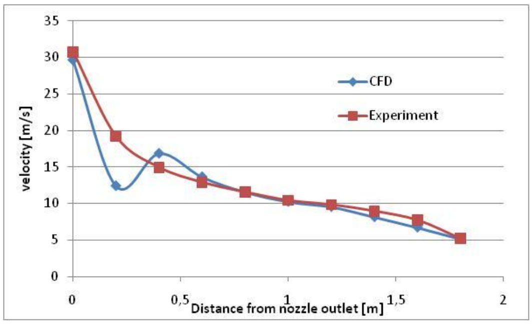


Figure 7 View of the CFD model

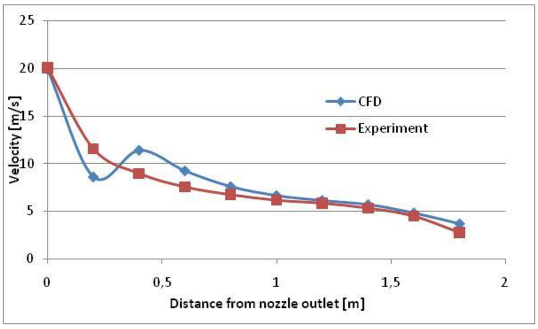
Displayed numerical simulations were conducted using ANSYS Fluent 14.5. software. What is more, RANS k-ε and LES turbulence model were applied in the calculations. Describing the picture: boundary conditions are defined as walls for the side, bottom and top of analysed domain. Additionally, the air curtain slot is defined as velocity inlet; upstream and downstream part of a model is pressure inlet. After the appropriateness of the numerical confirmation, various configurations, to check the behaviour of an air stream, have been investigated; for instance, diverse pressure differences and outlet velocities.. Concluding, results from the CFD calculations were compared with those from the experiment.

RESULTS

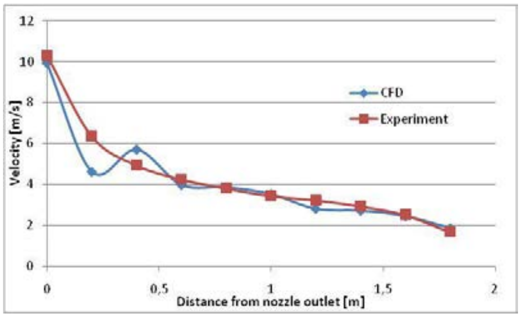
The effects of author's numerical calculations of free jet are in an extremely good agreement with the examined results from experiment, except the first part of the stream. As illustrated in Figure 8, the outcomes of the centreline velocity reveal immense similarity to those given in the experiment. In order to visualize the velocity profile, the velocity counters in cross-section of analytical domain are presented in figure 9 and 10 for both turbulence models.



a



b



c

Figure 8 Centreline Velocity
(I_0=5%, α=0°,
a)U=30 m/s, b) U=20 m/s, c)U= 30 m/s)

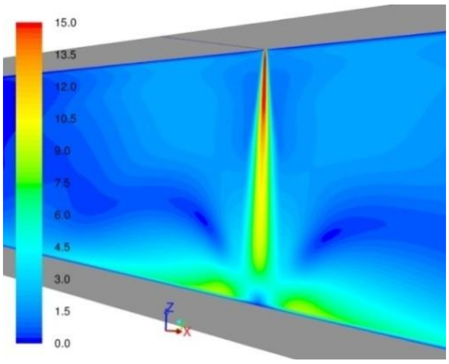


Figure 9 Velocity counters in cross-section of numerical domain (k- ε turbulence model)

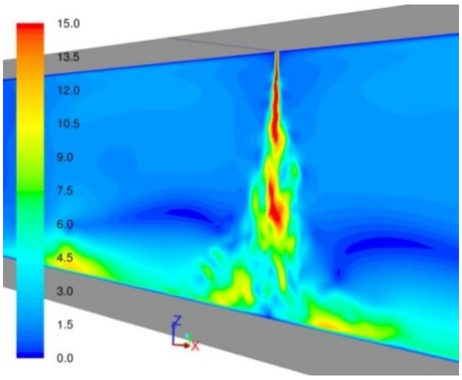


Figure 10 Velocity counters in cross-section of numerical domain (LES turbulence model)

An example of the idea of using air curtain as a separation of two zones - one full of smoke and one free of smoke is presented on a Figure 11.

Figure 11 Division of two zones using air curtain

An example visualisations of air curtain under the influence of lateral pressure are presented below on Figures 12 and 13, the curvature of an air stream is extremely corresponding for both turbulent models k-ε and LES. There is an essential difference in velocity profile but both of them are substantially similar to laser visualisations presented in literature.

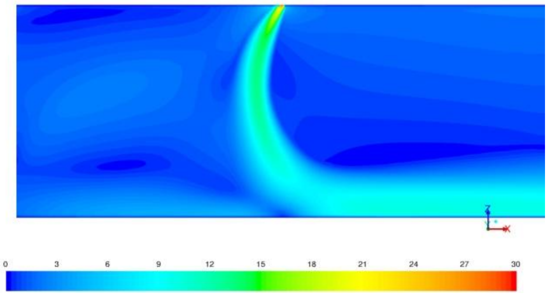


Figure 12 Velocity profile in cross section (k-ε turbulence model)

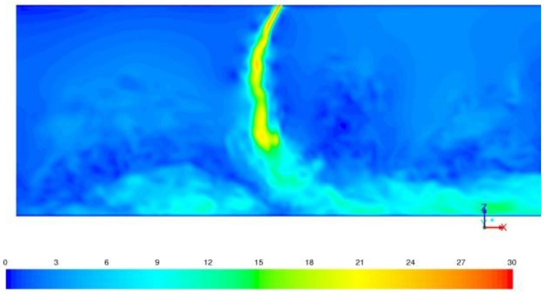


Figure 13 Velocity profile in cross section (LES turbulence model)

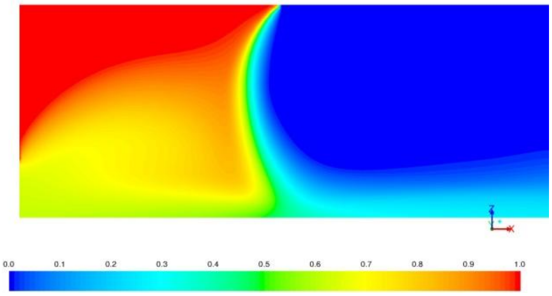


Figure 14 Mass fraction of tracer gas in cross section (k-ε turbulence model)

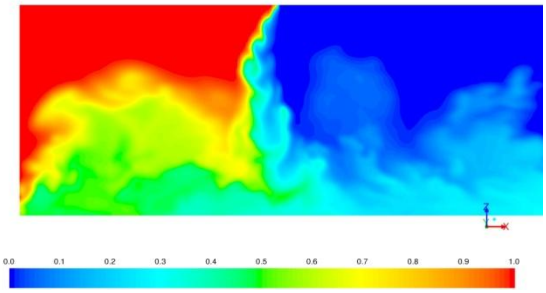


Figure 15 Mass fraction of tracer gas in cross section (LES turbulence model)

In Figure 14 and 15 a division of two separate zones of fluid is presented. One of them is full of smoke and the second one, on the opposite side of the air curtain, is almost free of smoke.

Moreover, in both settings of k-ε turbulence model and LES turbulence model, the amount of smoke which is transferred through the air curtain is significantly low. More detailed analysis of air curtain leakages will be done in farther research using a gas tracer decay method which will be implemented to CFD calculations.

CONCLUSION

The study presented in this paper was an attempt to demonstrate a possibility of using CFD methods which can help analyse all required parameters to check if an air curtain is properly designed. According to conducted simulations, it is crucial to declare that air curtains can be used as a division of smoke free zones in case of fire.

In addition, properly defined boundary conditions and chosen turbulent model have fundamental influence on received effects.

RANS k-ε turbulence model provides results especially quickly; however, the conclusions are not particularly similar to the reality and especially due to the fact that the impingent zone is modelled incorrectly. Parameters of the air curtain can be undersized. For more detailed analysis, it is better to use LES model with an exceptionally dense mesh because the model is very susceptible to a size of the mesh and can give false results if elements are oversized.

. In case of further studies, it is critical to define parameters of an air curtain which would be tight enough to completely separate two neighbouring zones. Additionally new calculations will be done with SAS turbulence model. Probably results from CFD calculations for different turbulence models with different sub-models will give better accuracy. In further studios there will be done full scale tests of air curtain which is influenced be pressure difference and high temperature. It is predict that full scale tests results will be very similar to dose from CFD calculations.

REFERENCES

Abramovitch G.N.: The theory of turbulent jets. Massachusetts M.I.T. press, 1963

ANSYS Fluent 13.0.0. Technical Documentation

Goyonnaud L., Solliec C.: Mass transfer analysis of an air curtain system, Transactions on Engineering Science vol. 18, 1998 WTT Press

Gugliermetti F., Santrapia L., Zori G.: Air curtain applied to fire smoke pollution control, Transactions on Ecology and the Environment vol. 66, 2003 WTT Press

Guyonnaund L., Solliec C., Dufresene de Virel M., Rey C.: Design of air curtains used for area confinement in tunnels, Experiments in Fluids 28 (2000) 377-384 Springer-Verlag 2000

Gupta S., Pavageau M., Elicer-Cortes J. C.: Cellular confinement of tunnel sections between two air curtains, Building and Environmant 42 (2007) 3352-3365

Hayes F. C., Stoecker W. F.: Heat Transfer Characteristics of the Air Curtain, ASHRAE Transactions No. 2120, 1969

Hayes F. C., Stoecker W. F.: Desig Data For Air Curtains, ASHRAE Transactions No. 2121, 196

Rajaratnam N.: Turbulent jets. Elsevier, Amsterdam, 1976

Schlichting: Boundary layer theory. Mc Graw-Hill Book, New York, 1968

Sztarbala G., Krajewski G.: Application of CAE in designing process of fire ventilation system based on jet fan system, EBECC London, UK, 2009

Krajewski G., Sztarbała G.: Air curtain used for separating smoke free zones in case of fire, 12th International Conference of Building Simulation 2011