Parametric CFD study of an air curtain for smoke confinement

L.-X. Yu[*1, 2], T. Beji[2], S. Ebrahim Zadeh[2], F. Liu[*1], B. Merci[2]

[1]Chongqing University, Faculty of Urban Construction and Environmental Engineering
[2]Ghent University – UGent, Dept. of Flow, Heat and Combustion Mechanics

Abstract
The CFD simulation of an air curtain for smoke confinement in case of fire is conducted by using Fire Dynamics Simulator (FDS 6.0.1). It is a working progress of preliminary simulation for preparation of small scale experiment. The set-up is a wind tunnel configuration. Special focus is given to the smoke flow field, jet velocity and temperature distribution in the protected area. Predicted ceiling jet properties are compared with analytical equations. Investigation of different jet velocities reveals that the smoke flow field in the wind tunnel is strongly influenced by the operation of air curtain. Jet velocity between 0.75m-1m/s is recommended for the study at hand.

1. Introduction

It is well-known that most fire deaths are caused by smoke inhalation. Several techniques have been developed in order to control smoke and remove heat generated by a fire, such as the pressurization of stairwells. However, large air supply volumes are required in this case. Therefore, air curtains could be a more efficient way of blocking smoke dispersion during fires[1].

The idea of aerodynamic sealing can be dated to the early twentieth century and was brought forward by Van Kennel in 1904[2]. In the past 50 years, the increased awareness for energy saving has led to a widespread use of air curtains, triggering theoretical and experimental research on their sealing ability[3]. One of the major applications of air curtains occur in preserving low temperatures in the refrigerated storage rooms[2]. The destabilizing factor of the air curtain is the stack effect caused by the temperature difference and thus density difference between the air inside and outside the room. However, in the context of fire, the transverse force of the fire-induced flow is much stronger than the natural convection infiltration through cold store entrances, due to a much higher temperature difference. Therefore, detailed studies must be carried out in order to address and evaluate the effectiveness of smoke confinement using an air curtain.

For the present work, the smoke flow field, jet velocity and temperature distribution in the protected area are analyzed using the CFD package FDS (Fire Dynamics Simulator), Version 6.0.1[4]. The aim of this study is to have a better understanding of the working process of an air curtain device (ACD) for smoke confinement.

2. General set-up of the simulations

A schematic view of the geometry of the test section is provided in Fig.1. The configuration is based on the test section of low speed wind tunnel called 'L2B' from Von Karman institute, where small-scale experiment will be conducted.

2.1 Dimensions and boundary conditions

The dimension of the wind tunnel is 35cm×35cm×200cm. The left and right end of the wind tunnel, marked in grey, are open to the outside by specifying them to be 'OPEN'[4]. The other four sides are 'solid' boundary conditions with the temperature fixed at ambient temperature (20°C), and are referred to as 'INERT'[4].

In order to resemble, as closely as possible, the real turbulent planar jet flow with a limited calculation cost, the air curtain flow is created through an inlet duct, marked in purple, with duct length equal to 1 hydraulic diameter of the inlet (4cm)[5]. The air curtain inlet, marked in blue, has a dimension of 35cm (length, L) × 2cm (width, W), resulting in an aspect ratio AR=L/W of 17.5, at 100cm away from the hot-smoke inlet. A power-law velocity profile[6], resemble the fully developed velocity profile, is imposed at the inlet. The profile has been determined from a preliminary calculation (not shown here) of fully developed velocity profile as obtained at the orifice exit with imposed top-hat velocity of 1m/s at the inlet of a 15 hydraulic diameter long inlet duct(60cm) [5]. Five maximal velocities of 0.5m/s, 0.75m/s, 1m/s, 1.5m/s and 2m/s are applied at the inlet. The synthetic eddy method (SEM)[7] is applied to the air curtain as specification of turbulent inlet boundary condition for a more realistic turbulent jet flow simulation . The artificial eddies are implemented in FDS by specification of the number of eddies (N_eddy=1120), the characteristic (integral) length scale (L_eddy=0.0028m) and the amplitude of turbulent fluctuations (VEL_RMS=0.05, 0.075, 0.1, 0.15 and 0.2m/s, I = 10%). The detailed set up of air curtain inlet and the reasons for the chosen parameters of SEM are investigated in a separate study, the reader is referred to the FDS manual[4] and Jarrin's thesis[7] for detailed information.

The round hot-smoke inlet, with diameter of 5cm, flush with the floor and marked in red, has a velocity inlet boundary condition. It is the source of 'hot smoke'. Hot air with temperature of 300°C, is injected at the inlet with uniform velocity of 1m/s. This is equivalent to a fire

* Corresponding author: longxing.yu@ugent.be; drliufang@126.com
Proceedings of the 2nd IAFSS European Symposium of Fire Safety Science

source with heat release rate of 0.67 kW. The Froude number at the inlet is 1.43.

Since more detailed simulation of the fire smoke is out of the scope of the present study, simplification at the inlet boundary condition is made, i.e., top-hat velocity profile is imposed and no synthetic eddy method is applied. However, the accuracy of simulation result is sufficient. As shown in section 3.1, well resolved ceiling jet is comparable to the analytical correlations.

2.2 Grid resolution and turbulence models

Uniform orthogonal structured hexahedral cells (cubic cells) with grid resolution of W/Δ =8 (Δ=0.25cm) are used in all simulations. This results in 16 cells across the hydraulic diameter of the air curtain inlet, 20 cells across the hydraulic diameter of the hot-smoke inlet and in total 15,904,000 cells within the computational domain. Parallel calculations with 32 processes in total are adopted. It has been proven to be sufficiently fine for the simulations at hand[5].

The dynamic Smagorinsky turbulence model is applied in all the simulations[4].

2.3 Test cases and measurements

Six simulation cases have been carried out. The first case focuses on the simulation of the ceiling jet and only consists of hot-smoke injection. As mentioned above, the other five cases focus on the effect of jet velocity. Five maximal velocities of 0.5m/s, 0.75m/s, 1m/s, 1.5m/s and 2m/s are applied to the air curtain inlet. Based on the reality and for a better knowledge of air curtain operation in case of fire, the air curtain operates 10 seconds after the fire happened (hot-smoke injection).

Fig.1 shows the computational domain and line measurements in CFD as well. Four line measurements in the symmetry plane recording the ceiling jet velocity, and temperature profiles, are marked in green. They are distributed upstream the air curtain inlet at intervals of 20cm.

All the mean values are retrieved as output from the simulations, averaging results over the last 10 seconds of the total of 30 seconds calculation time, which is sufficient for the flow to reach steady-state. Note that 30 seconds corresponds to approximately 6 ceiling jet flow-through times through the wind tunnel.

3. Results and discussion

3.1 Ceiling jet

When a plume impinges on a ceiling, hot gases spread out radially beneath the ceiling, driven by the buoyancy of the hot combustion products from the plume. This is the so-called ceiling jet[8]. Research studies designed to quantify the temperatures and velocities of this initial spreading flow have been conducted since the 1950s.

Alpert[8] provided simple correlations to quantify the maximum gas temperature and velocity at a given position in a ceiling jet flow produced by a steady fire. These correlations are widely used in hazard analysis calculations.

Heskestad[8] developed nondimensional ceiling jet correlations for maximum ceiling jet excess temperature and velocity based on alcohol pool-fire tests performed at the U.K. Fire Research Station in the 1950s.

Figure 2 reveals that the present simulation result shows a good agreement with the correlation in terms of excess velocity, particularly with Heskestad's correction [8]. However, for the ceiling jet temperature, the simulation shows a lower decay rate compared with those two correlations. This is because those correlations are developed based on steady-state fires under unconfined ceilings[8]. During the experiment, the large compartment size did not allow for the development of a significant hot gas layer. However, for the configuration at hand, the hot gas layer develops relatively fast, and temperatures at the ceiling are affected by the ceiling jet as well as the accumulating hot gas layer. Thus the correlation tends to underpredict the temperatures and shows a faster decay because it is not accounting for the development of the hot gas layer. This is in line with [9].

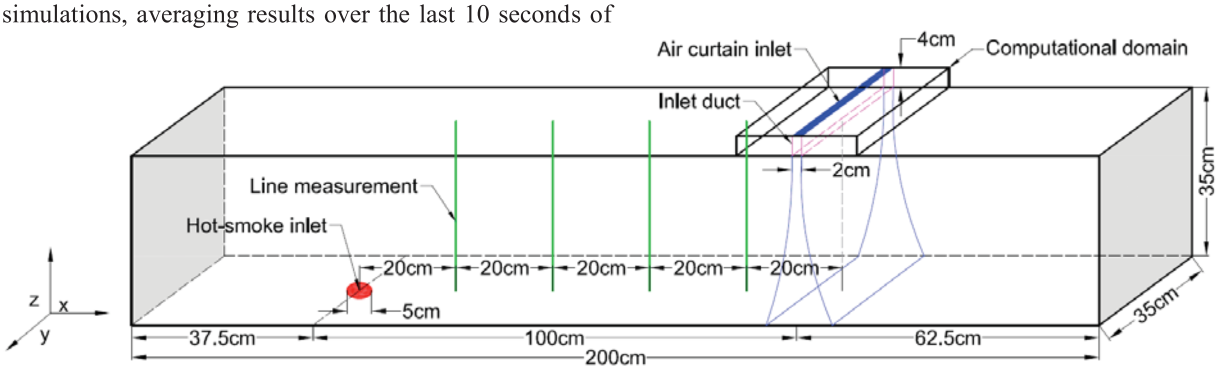

Fig.1 Schematic of the test section of low speed wind tunnel 'L2B'

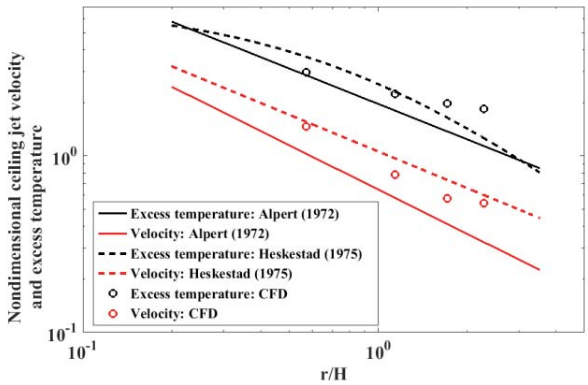

Fig.2 Comparison of dimensionless correlations for maximum ceiling jet temperatures (black) and velocities (red) produced by steady fires. Solid line: Alpert[8]; dashed line: Heskestad[8]; symbol: CFD results.

3.2 Smoke flow field with interaction of air curtain

In this sub-section, the smoke flow fields, especially the space between the fire and air curtain, with different jet velocities are discussed. It is instructive to carefully examine the effect of the air curtain on the air movement inside the wind tunnel. Obviously, the air movement in the wind tunnel will inevitably and simultaneously affect the smoke spread and the effectiveness of air curtain.

The up four figure of Figure 3 shows the vertical temperature profiles, with and without air curtain operation, at different locations in the symmetry plane of wind tunnel. The jet velocity of 1m/s is chosen here for complete sealing and velocity of 0m/s refers to the absence of an air curtain. It shows that, with successful air curtain blockage, even though the smoke is flowing out through the left opening of the wind tunnel, the smoke still accumulates to a certain extent in between the fire and the air curtain, causing the smoke layer to descend by approximately 0.05m (15% of ceiling height). It reveals very well the sealing effect of air curtain. However, the maximal ceiling jet temperature shows very small influence from the air curtain. The cooling effect by the injected cold air (20℃) from air curtain can only be slightly noticed at the near place (Δx=0.8m). As expected, the sealing effect is the strongest near the air curtain (Δx=0.8m), but even close to the source (Δx=0.2m) the impact of the air curtain is substantial.

Moreover, the smoke flow field is strongly affected by the air curtain operation. The bottom four figure of Figure 3 shows the profiles for horizontal velocity in the x direction, with and without air curtain operation, at different locations in the symmetry plane of wind tunnel. Without the air curtain operation, all the velocity profiles (black lines) have an 'Inverted S-shaped', i.e., the profiles have a positive value at the top and negative value at the bottom.

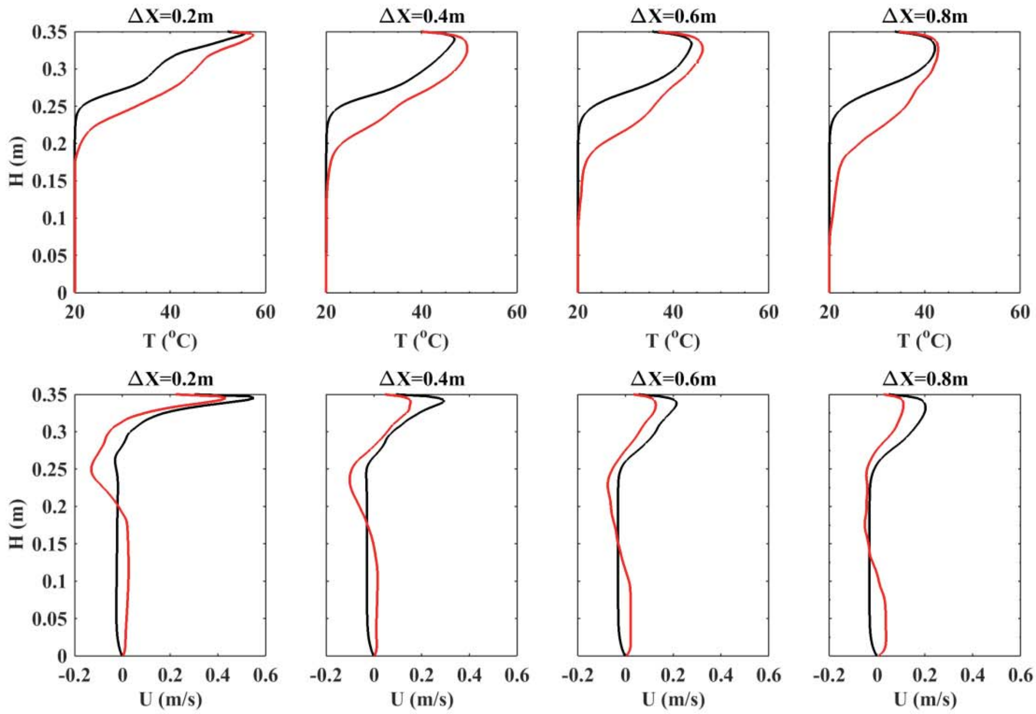

Fig.3 Vertical profiles of mean temperature (up) and mean horizontal velocity (bottom) at different locations in the symmetry plane of the wind tunnel. Black lines: without air curtain operation, V=0m/s; Red lines: with air curtain operation, V=1m/s.

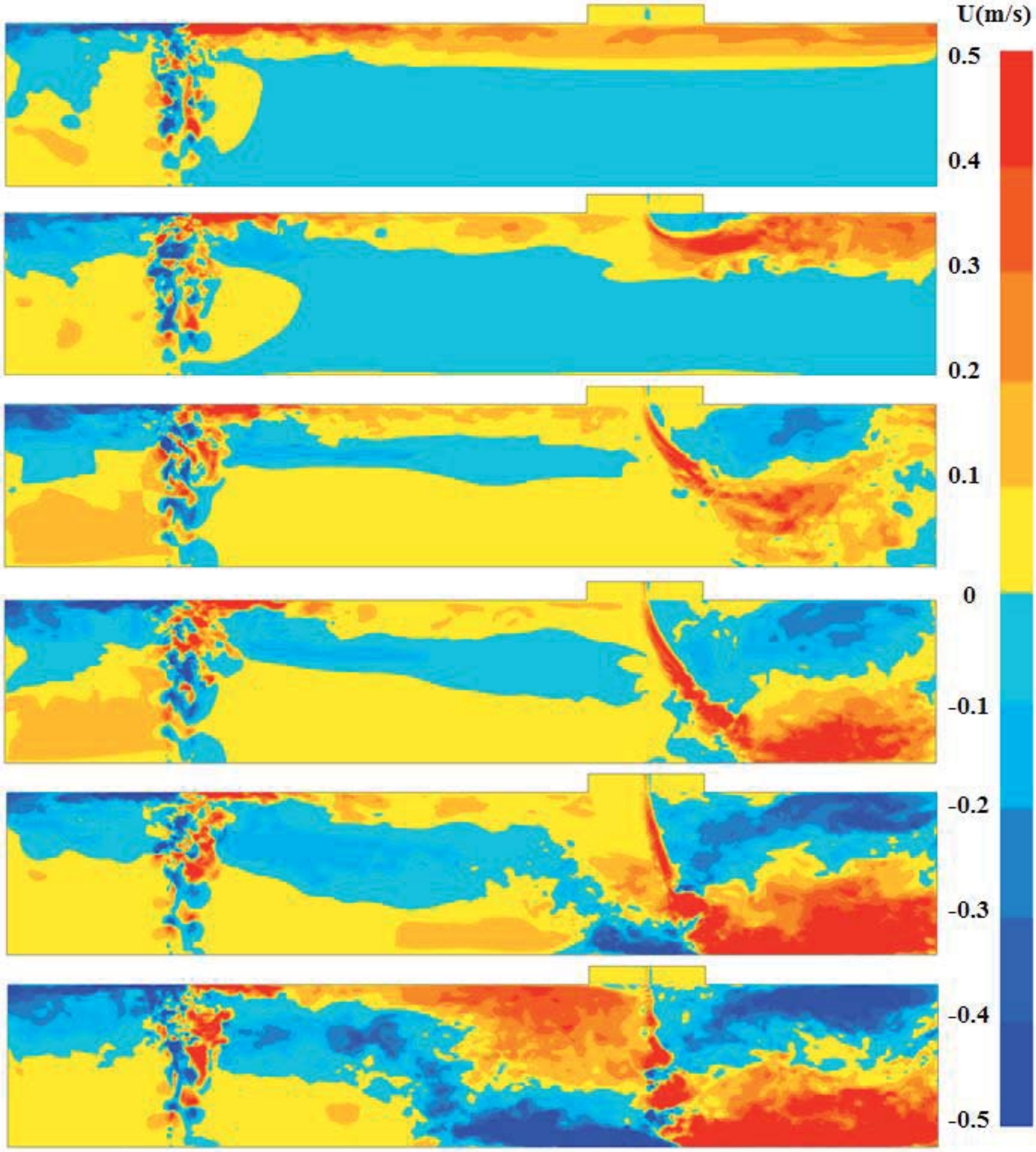

Fig.4 Snap shot of horizontal velocity contour at t=25s in the symmetry plane of the wind tunnel. U>0 represents flow to the right; U<0 represents flow to the left. From top to bottom: V=0m/s, V=0.5m/s, V=0.75m/s, V=1m/s, V=1.5m/s and V=2.0m/s.

This is because the smoke flow field at the right side of the fire can be divided into 'two zones' with different flow directions (see top figure of Fig.4), i.e., the hot upper layer moves to the right ($U>0$) and the cold bottom layer moves to the left ($U<0$) due to the buoyancy and entrainment. However, with the air curtain in operation, the smoke flow field will be stratified into 'three zones' (from the third figure of Fig.4 onward). The upper and lower layer moving to the right ($U>0$) and the middle layer moving to the left ($U<0$) is observed.

As mentioned above, Figure 4 shows snap shots of horizontal velocity layout at t=25s in the symmetry plane of wind tunnel, with jet velocity varying from 0 to 2m/s. As the jet velocity increases from 0 to 2m/s, the flow field evolves from 'two zones' (top figure of Fig.4) to 'three zones' (bottom figure of Fig.4). This is due to the momentum of the air curtain. However, as the air curtain velocity increases from 0.75m/s to 2m/s the lower layer region decreases and the middle layer region is extended (from the third figure of Fig.4 onward). Less deflection of the air curtain is clearly observed with a higher jet velocity.

3.3 Influence of air curtain on the temperature field

In this sub-section, the temperature fields, especially the protected area at downstream of air curtain, with different jet velocities are discussed. It is obvious that the temperature distribution at the protected area strongly depends on the jet velocity. Fig.5 shows the mean temperature contour of the air curtain region and the protected area, i.e., between $1.25m<X<2m$ (the air curtain locates at X=1.375m). It shows that the temperature at the protected area is substantially reduced when the air curtain velocity is higher or equal to V=0.75m/s. For the latter cases (i.e. $V \geq 0.75$ m/s) the temperature fields are only slightly different. On the contrary, as mentioned in section 3.2, as the jet velocity increases, more back flow with fresh air will move to the fire source. Thus, jet velocity between 0.75m-1m/s is recommended for the study at hand.

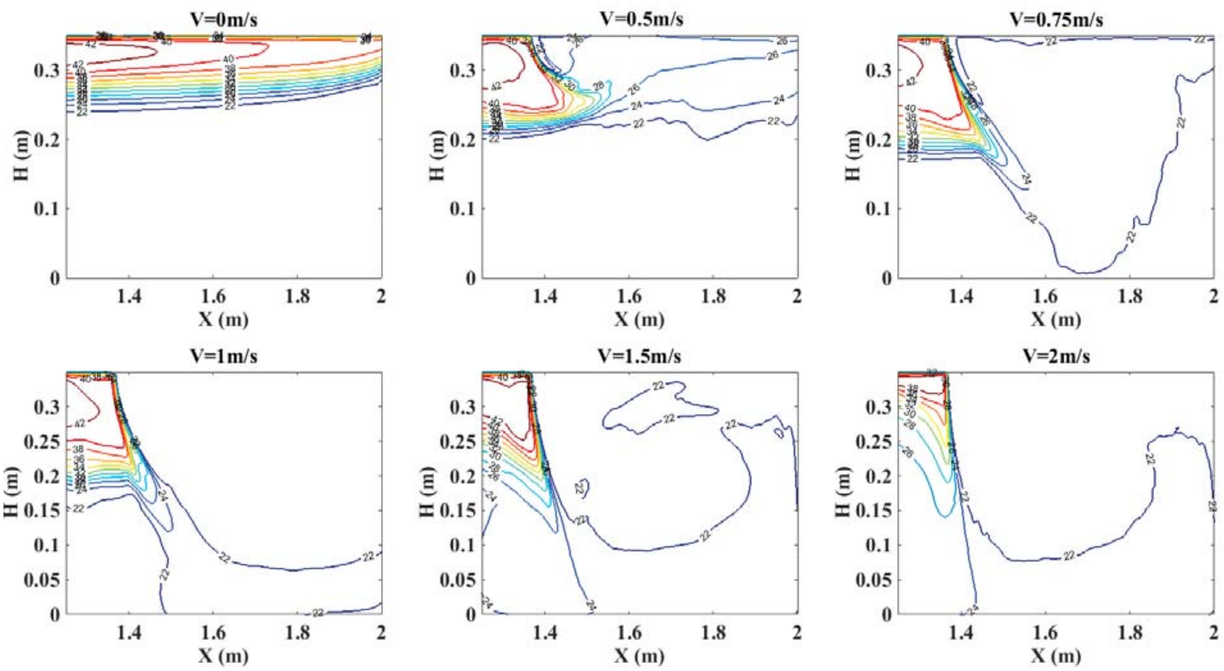

Fig.5 Local mean temperature contour ($1.25m<X<2m$) in the symmetry plane of the wind tunnel. From top to bottom and left to right: V=0m/s, V=0.5m/s, V=0.75m/s, V=1m/s, V=1.5m/s and V=2.0m/s.

4. Conclusions

Preliminary FDS simulation of air curtain flows in smoke confinement has been performed as a preparation for small scale experiment. A goal of better understands of the smoke flow field, jet velocity and temperature distribution in the protected area for the configuration at hand is reached. The key findings of the present work are as follows.

The present simulation of ceiling jet shows a good agreement in the maximal velocity with Heskestad's correlation. However, the corrections tend to underpredict the temperatures and show a faster decay because it is not accounting for the development of the hot gas layer.

The smoke flow field in wind tunnel is strongly influenced and determined by the operation of air curtain. Two flow zones are found in a failed ACD while three flow zones are found in a successful set-up of ACD.

Moreover, the pattern of the three flow zones changes as the air curtain velocity increases, i.e., the lower layer region decreases and the upper and middle layer regions increase.

Designing the air curtain, care must be taken when choosing the jet velocity. Jet velocity between 0.75m-1m/s is recommended for the study at hand.

Acknowledgements

This research has been conducted at Ghent University, supported by the State Scholarship Fund of China under grant No.201306050081 and Chongqing Graduate Student Research Innovation Project under grant No.CYB14031.

Dr. Tarek Beji is Postdoctoral Fellow of the Fund of Scientific Research – Flanders (Belgium) (FWO-Vlaanderen)

The computational resources (Stevin Supercomputer Infrastructure) and services used in this work were provided by the VSC (Flemish Supercomputer Center), funded by Ghent University, the Hercules Foundation and the Flemish Government – department EWI.

Reference

[1] Moriyama S, Smoke Control System for High-Rise Buildings in Japan, Journal of Disaster Research, 2011;6: 551-52.

[2] Foster AM. CFD Optimization of Air Movement through Doorways in Refrigerated Rooms. In. CFD Optimization of Air Movement through Doorways in Refrigerated Rooms. CRC Press, 2007, pp. 167-93.

[3] Frank D, Linden PF, The effectiveness of an air curtain in the doorway of a ventilated building, Journal of Fluid Mechanics, 2014;756: 130-64.

[4] McGrattan KB, Hostikka S, Floyd J, Fire Dynamics Simulator, User's Guide, NIST special publication, 2013;1019.

[5] Yu L-x, Liu F, Merci B, Analysis of the impact of the inlet boundary conditions in FDS results for air curtain flows in the near-field region, Ninth Mediterranean Combustion Symposium, Rhodes, Greece, 2015 (accepted).

[6] Munson BR, Young DF, Okiishi TH, Fundamentals of fluid mechanics, New York, 1990.

[7] Jarrin N, Prosser R. Synthetic inflow boundary conditions for the numerical simulation of turbulence. University of Manchester, 2008.

[8] Bryan JL, SFPE handbook of fire protection engineering, 3rd ed., National Fire Protection Assosiation, USA, 2002.

[9] Overholt KJ, Verification and Validation of Commonly Used Empirical Correlations for Fire Scenarios, 2014.