Flow behaviour of the exchange flow through a ceiling vent in natural convection: A numerical approach using CALIF$^{3}$S-ISIS CFD software

Patricio BECERRA BARRIOS$^{1,3}$, Hugues PRETREL$^{2}$, Samuel VAUX$^{2}$ and Olivier VAUQUELIN$^{1}$

$^{1}$Aix-Marseille Université (AMU), Laboratoire IUSTI, Laboratoire ETiC, France
$^{2}$Institut de Radioprotection et de Sûreté Nucléaire (IRSN), Laboratoire ETiC, France
$^{3}$School of Mechanical Engineering, University of Costa Rica (UCR), Costa Rica

patricio.becerra@ucr.ac.cr

ABSTRACT

In the fire safety context, the exchange flow through a horizontal opening connecting two superposed zones receives a particular interest for developing predictive models and validation of computational codes. Its random behaviour over time could influence the fire source and the vertical spread of the fire smoke. This work is part of a research program developed at the French "Institut de Radioprotection et de Sûreté Nucléaire" (IRSN), and its main objective is to bring new knowledge on this subject by carrying out a numerical study with the computational open-source software named CALIF$^{3}$S-ISIS developed by IRSN. The configuration of the simulations is that of an experimental reduced-scale set-up, on which data have been obtained and could be used for validation. Large Eddy Simulation (LES) were performed (variation of wall boundary conditions, air injection temperatures, vent diameter and vent mesh refinement), and the bidirectional flow is properly estimated in comparison with the experiment. Particularly, it was determined that the increment of the geometric ratio $L/D$ of the vent of diameter $D$ and thickness $L$, also leads to an increment of a dimensionless exchange flow rate Froude number at the vent. However, a discrepancy was observed with the model proposed by Epstein based on his experiments in natural convection. The disagreement between the simulation results and the experiments is attributed principally to differences on the vertical temperature stratification of the lower room. Finally, it was investigated the influence of the vertical location within the vent over the mean flow behaviour through different levels of the vent, specifically an exchange of the downward and upward flow organization within the opening.

KEYWORDS: bidirectional flow, horizontal vent, CALIF$^{3}$S-ISIS CFD code
NOMENCLATURE LISTING

\begin{align*}
D & \quad \text{diameter (m)} \\
Fr & \quad \text{Froude number (-)} \\
g & \quad \text{gravitational acceleration (m/s}^2) \\
L & \quad \text{height of the vent (m)} \\
Q_v & \quad \text{volume flow rate (m}^3/\text{s}) \\
Q_m & \quad \text{mass flow rate (kg/s)} \\
T & \quad \text{temperature (K)} \\
T_s & \quad \text{simulation time (s)} \\
t & \quad \text{time (s)} \\
z & \quad \text{vertical distance above floor level (m)} \\
\rho & \quad \text{density (kg/m}^3) \\
\bar{\rho} & \quad \text{average density (kg/m}^3) \\
\rho_{+} & \quad \text{upward flow rate} \\
\rho_{-} & \quad \text{downward flow rate} \\
\infty & \quad \text{ambient condition}
\end{align*}

INTRODUCTION

Smoke flow through openings between different areas of a building represents a challenge to safety during a compartment fire. Its mass and energy transfer could have a significant impact on the control of contaminant particles and the spread of fire and smoke. In this study, we draw attention to the buoyancy-driven exchange flow through an horizontal opening in natural convection due to a confined fire source in a compartment, see Figure 1.

The earliest studies dealing with this problem were carried out by Brown [1] and Mercer & Thompson [2]. Brown [1] used air as working fluid and interpreted the exchange flow as a transfer phenomena and concluded that the exchange flow rates increase with the increase of the geometrical ratio $L/D$. Mercer & Thompson [2] using brine solutions and water in inclined and vertical pipes, reported the opposite behaviour observed by Brown and suggested a maximum exchange flow rate at an $L/D$ between 0.66 and 3.5. Epstein [3] completed their works using water-brine as fluid and studied in detail the effect of the vent aspect ratio. Epstein proposed a correlation expressing the volume exchange flow $Q_v$ in the form of a dimensionless Froude number at the vent of diameter $D$ and thickness $L$, $Fr = \frac{Q_v}{\sqrt{gD^5 \Delta \rho / \bar{\rho}}}$, where $\Delta \rho$ is the density difference between the top and the bottom of the opening, as a function of the geometric ratio $L/D$ of the vent. He identified the exchange flow behaviour in four regimes from an oscillatory regime to a turbulent diffusion flow. Li [4] conducted full-scale experiments with fire using the laser Doppler velocimetry (LDV) technique to measure the flow at the vent. Li also performed CFD calculations using the standard $\kappa - \epsilon$ turbulence model and the Large Eddy Simulation (LES) technique. However, Li [4] concluded that only simulations using LES were in agreement with his experiments. Recently, Varrall [5] investigated the volume exchange flow using the stereoscopic particle image velocimetry (SPIV) technique as a non-intrusive measurement way. Varrall measured the velocity fields at the vent in a thermal steady state, after heating the air in the lower compartment. He determined an average flow organization with the upward flow occupying an egg-shaped central area and the downward flow occupying the parietal areas.

\begin{figure}[h]
\centering
\includegraphics[width=0.5\textwidth]{fig1.png}
\caption{Bidirectional flow exchange through the vent in an enclosure fire. Geometrical configuration.}
\end{figure}

Few numerical studies using the Computational Fluid Dynamics (CFD) focused on that problem. Spall & Anderson [6] investigated the exchange flow through a horizontal vent for a thin partition and indicated the presence of two dominating frequencies in the oscillatory regime. Harrison & Spall [7] studied the effects of the partition thickness to complete the work of Spall & Anderson [6] in the oscillatory flow regime. Sleiti [8] studied the effects of the geometrical ratio of the vent on the oscillatory regime using FLUENT software. His results indicated that the exchange flow and frequency increased when the aspect ratio decreased from 1.0 to 0.5, and as the vent ratio increased from 1.0 to 2.0, the exchange flow decreased and low frequency regime appeared.

The case examined herein is part of the study carried out at the French "Institut de Radioprotection et de Sûreté Nucléaire" (IRSN) in the context of a large research project program [9]. Specifically, this work focuses on the behaviour of the exchange flow at the horizontal vent observed in full-scale fire experiments. The aims of the present study is to evaluate the capability of the CALIF$^3$-ISIS open-source CFD software developed by
the IRSN to predict the bidirectional exchange flow through the vent and to investigate the flow inside the vent. The turbulence modelling used to simulate such flows relies on the LES technique. The simulated flows will be compared with the tendency of the function $Fr(L/D)$ of literature data, in which the exchange flow through the vent is a consequence of the density difference between the compartments. Finally, the experimental data obtained by Varrall [5] will be used for comparison with the numerical simulations based on the mean velocity field.

CASE STUDY

The case configuration (Figure 2a) is an experimental reduced scale set-up, which consists in two superposed compartments connected by a circular horizontal opening. The internal dimensions of the lower compartment are 1m x 1m x 1.5m, and the dimensions of the upper one are 1m x 1m x 1m. The length or height $L$ of the partition between the compartments, which receive the vent, is 38mm thick. The steel framework allows different types of wall materials: glass, steel and insulating material. The fire source in the lower compartment was simulated experimentally by one electrical resistor of 2 kW electrical power. A semi-elliptical fine wire mesh was located to reduce the ceiling jet effect on the flow through the horizontal opening produced by the thermal heat source.

The simulations presented hereafter were performed using the CALIF$^3$S-ISIS software (version 4.2) dedicated to the simulations of fire in confined and mechanically ventilated compartments [10]. The computational domain was based on the experimental set-up mentioned above. Four vent diameters were used: 127mm, 152mm, 191mm and 260mm. The buoyancy source was simulated by forcing a jet of hot air vertically upward into the lower compartment instead of hot plate as experimentally. The source was a 82 mm diameter nozzle located at 200 mm above the floor level. The hot air was injected at 0.43 m/s to maintain a Reynolds number close to 2000 at the injection. Two temperatures of hot air were tested: 363 K and 423 K to simulate the experimental vertical stratification temperature profile. The experimental vertical gradient profile was about 30°C/m at the steady state. Two wall boundary conditions were used: conduction and adiabatic walls. An exhaust on the opposite side of the nozzle at floor level was included to impose a mass flow conservation in the lower compartment, due to mass injection of the hot air by the nozzle. Also a screen to minimize the impact of the ceiling jet effect through the exchange flow was added to evaluate its influence over the exchange flow. Turbulence was modelled using the LES technique and WALE (Wall Adapting Local Eddy viscosity) subgrid-scale model. The Favre-filtered continuity, momentum and energy transport equations are solved. Time discretization is performed by a fractional step algorithm decoupling balance equations for the transport of energy and Navier-Stokes equations which are solved by a pressure correction technique. The horizontal opening was divided in three height levels: $z = 100.0 \, \text{cm}$, $z = 101.9 \, \text{cm}$ and $z = 103.8 \, \text{cm}$, to investigate the flow behaviour inside the opening. The lowest slice coincides with the ceiling level of the lower compartment. Note that a convergence study has been performed in order to validate the refinement of the different meshes (M1: 1.2 million cells, M2: 1.9 million cells, M3: 2.7 million cells, M4: 3.5 million cells). Different vertical and horizontal grid spacings were tested (both in the rooms and within the vent). We considered convergence was reached as soon as the mean mass flow evolution at the vent was less than 5% than the reference value obtained with the finest grid, M4. A structured Cartesian mesh was used with a mesh refinement applied at the vent, from 25 to 100 thousand cells. A schematic mesh of the configuration is shown in Figure 2b.

![Experimental set-up.](image1.png)

![Schematic mesh.](image2.png)

Fig. 2. Configuration of the experiment (a) and the corresponding mesh (b) for the simulation.
RESULTS

Features of the flow at the vent

The capability of the code to assess the bidirectional flow exchange at the vent is presented in Figure 3, where the net flow mass rate tends to zero after 300 s of simulation. Downward flow is plotted on negative values for visualisation purposes.

![Fig. 3. Mean mass flow evolution for D = 260 mm at z = 101.9 cm.](image)

Figure 4 presents instantaneous velocity fields at different simulation times. The positive values correspond to the upward flow and negative values to the downward flow. These figures show as observed experimentally by Varrall [5] the randomly distribution of the flow and there is no specific area for the upward or downward flow over time.

![Fig. 4. Instantaneous velocity field for D = 191 mm at z = 101.9 cm.](image)

To understand the flow pattern on average over the simulations, Varrall’s procedure [5] was used to determine the mean flow at the vent for the overall simulation time $T$. The simulated mean velocity fields for diameters $D = 191 \text{ mm}$ (Figure 5a) and $D = 260 \text{ mm}$ (Figure 5c) both at $T = 300 \text{ s}$ were compared with their respective experimental results (Figure 5b and Figure 5d). The calculations of the average velocity field for both diameters showed an organization of the upward flow at the centre of the vent, while the downward flow was constrained to the periphery close to the edge of the vent with a shear zone between them, in which the mean velocity was null.

![Fig. 5. Mean velocity field comparative: (a) and (b) for D = 191 mm, (c) and (d) for D = 260 mm at z = 103.8 cm.](image)
Geometric ratio $L/D$ effect and boundary conditions

It was observed that the Froude number $Fr$, expressed as Epstein [3], increases with the vent ratio for all the experiments simulated in the range of $0.1 \leq L/D \leq 0.5$. This is the case for the four geometric ratios simulated as shown in Figure 6a. This figure also presents a comparison between the simulations and some experimentally results available in the literature. The final cases simulated ($L/D = 0.15$ and $L/D = 0.20$) with higher refinement at the vent, use of the screen and conduction at walls, were closer to Epstein’s correlation.

For the two wall boundary conditions used, the thermal stratification conditions observed experimentally in the lower compartment could not be maintained in steady condition, even by imposing a temperature profile at the start time in the lower room, as shown in Figure 6b. For the case using adiabatic walls, the vertical temperature profile became also quasi-constant for the entire height of the lower compartment by the end of the simulations.

The rise of the hot air injection temperature at the source between $363 \, K$ and $423 \, K$ increased about $5 \, K$ the Froude number at the vent, use of the screen and conduction at walls, were closer to Epstein’s correlation.

For the two wall boundary conditions used, the thermal stratification conditions observed experimentally in the lower compartment could not be maintained in steady condition, even by imposing a temperature profile at the start time in the lower room, as shown in Figure 6b. For the case using adiabatic walls, the vertical temperature profile became also quasi-constant for the entire height of the lower compartment by the end of the simulations.

The rise of the hot air injection temperature at the source between $363 \, K$ and $423 \, K$ increased about $5 \, K$ the Froude number at the vent, use of the screen and conduction at walls, were closer to Epstein’s correlation. For that reason, the lower Froude number at the vent increased about $7\%$ moving away from Epstein’s correlation. For that reason, the lower vent remained.

Fig. 6. (a) Evolution of the experimental and numerical Froude number at the vent as a function of the ratio $L/D$. (b) Vertical temperature profile evolution for $D = 191 \, mm$ ($L/D = 0.20$), conduction and $T_{inj} = 363 \, K$.

Effect of vertical location within the vent

Figure 7 presents the influence of the horizontal slice height over the mean velocity field through the vent for two diameters and the same simulation time $T$. These velocity fields were not possible to obtain experimentally inside the vent because SPIV measurements were done at the top level of the vent. For both diameters, the lower level showed an opposite organization of the flow in comparison with the top level. At the lower level, the downward cold air occupied the central area while the upward hot air was observed close to the parietal zone. It has been observed thus an exchange of the flow pattern between the extreme levels of the vent. This behaviour of ‘corkscrew’ motion in vertical pipes have been observed in the core-annular flow of oil and water (see photos in Chap. VII [11]).

Fig. 7. Height influence for $D = 191 \, mm$ (a,b and c) and for $D = 260 \, mm$ (d,e and f), $T = 300 \, s$. 

\[ F_{r} = \frac{V_{inj}}{\sqrt{gD}} \]

\[ T \]
CONCLUSION

In this study, numerical simulations (based on LES approach) of the flow at a vent connecting two superposed 'zones' of different density show the existence of a bidirectional flow, as observed experimentally. Four simulations were performed to examine the influence of the $L/D$ vent ratio. It was observed that the increment of the aspect ratio $L/D$ results in an increment of the Froude number. Two wall boundary conditions, conduction and adiabatic walls, and two hot air injection temperatures were employed in the simulations. A screen to limit the ceiling jet flow produced by the source was evaluated. Simulations with conduction at walls, a fine mesh at the vent and the use of the screen, showed results closer to Epstein's correlation. However, a discrepancy was observed with the model proposed by Epstein based on his experiments in natural convention. The disagreement between the simulation results and the experiments might be attributed principally to differences on the vertical temperature stratification of the lower compartment. Finally, an exchange of the flow organization between the upward and downward flow within the vent levels was observed, following a 'corkscrew' motion in the thickness of the horizontal opening.

REFERENCES

[1] Brown, W. G. (1962) Natural convection through rectangular openings in partitions – 2, *International Journal of Heat and Mass Transfer* 5: 869–878, doi: 10.1016/0017-9310(62)90185-0.

[2] Mercer, A. and H. Thompson (1975) An experimental investigation of some further aspects of the buoyancy-driven exchange flow between carbon dioxide and air following a depressurization accident in a Magnox reactor, *Journal of the British Nuclear Energy Society* 14: 327–334.

[3] Epstein, M. (1988) Buoyancy-driven exchange flow through small openings in horizontal partitions, *Journal of Heat Transfer* 110: 885–893, doi: 10.1115/1.3250589.

[4] Li, Z. (2007) “Characteristics of buoyancy driven natural ventilation through horizontal openings”, PhD thesis, Aalborg University.

[5] Varrall, K. et al. (2016) Stereoscopic particle image velocimetry investigation of the bidirectional natural convection flow through a horizontal vent, *Fire Technology* 52: 2027–2041, doi: 10.1007/s10694-016-0593-3.

[6] Spall, R.E. and E.A. Anderson (1999) A numerical study of buoyant, pulsating exchange flows through a vent in a thin horizontal partition, *Numerical Heat Transfer: Part A: Applications* 36: 263–272, doi: 10.1080/104077899274750.

[7] Harrison, R.P. and R.E. Spall (2003) The effects of partition thickness on buoyant exchange flow through a horizontal opening, *Numerical Heat Transfer: Part A: Applications* 44: 451–462, doi: 10.1080/1040778032000122190.

[8] Sleiti, A.K. (2008) Effect of vent aspect ratio on unsteady laminar buoyant flow through rectangular vents in large enclosures, *International Journal of Heat and Mass Transfer* 51: 4850–4861, doi: 10.1016/j.ijheatmasstransfer.2008.02.027.

[9] Prétrel, H. et al. (2017) Experimental study based on large-scale smoke propagation fire tests through a horizontal opening connecting two mechanically ventilated compartments, *Fire Safety Journal* 90: 28–43, doi: 10.1016/j.firesaf.2017.02.007.

[10] Suard, S. et al. (2011) Verification and validation of a CFD model for simulations of large-scale compartment fires, *Nuclear Engineering and Design* 241: 3645–3657, doi: 10.1080/104077899274750.

[11] Joseph, D.D. and Y.Y. Renardy (1993) *Fundamentals of Two-Fluid Dynamics, Part II: Lubricated Transport, Drops and Miscible Liquids*, vol. 4, Springer Science & Business Media, doi: 10.1007/978-1-4615-7061-5.