NUMERICAL STUDY ON THE INFLUENCE OF WIND AND THERMAL STACK ON STREET CANYON AIRFLOW PATTERN
Abel Tablada*, Staf Roels*
*Katholieke Universiteit Leuven, Laboratory of Building Physics, Heverlee, Belgium.

Abstract
Building ventilation potential and pollutant dispersion are greatly affected by the airflow pattern in street canyons. The airflow inside a street canyon can be the result of wind washing and/or buoyancy due to solar radiation. The objective of this study is to analyse, through numerical simulations, the relative importance of wind versus thermal stack on the airflow pattern. 25 cases with different Froude numbers (Fr) are analysed for a square street canyon. The results show that for Fr < 30 a secondary stack-induced vortex develops along the heated wall. A weak secondary vortex near the ground is created at \( Fr = 6 - 7 \).

Key words: buoyancy, urban ventilation, Froude number, computational fluid dynamics

1. INTRODUCTION
Building ventilation potential and pollutant dispersion are greatly affected by the airflow pattern in street canyons. Two main reasons for airflow in a street canyon exist: wind-induced and buoyancy due to solar radiation on one of the facades and/or ground surface. During the last decades several studies have focused on measuring and simulating airflow inside street canyons under isothermal conditions. The main airflow patterns have been described according to canyon's geometrical characteristics [Oke, 1988]. However, more recent studies have extended the research towards the influence, on the airflow pattern and pollutant dispersion, of the incident solar radiation on the canyon walls and ground surfaces [Sini et al., 1996; Louka et al., 2000; Kovar-Panskus et al., 2002; Xie et al., 2005; Xie et al., 2007 and Niachou et al., 2008].

The objective of this study is to analyze the relative importance of wind versus thermal stack on the airflow pattern. This is done by analyzing the airflow pattern in a square canyon by Computational Fluids Dynamics (CFD) for the Kovar-Panskus et al. (2002) experimental configuration. This knowledge is important to further assess the street canyon ventilation potential, the possible shading strategies on building facades and the influence of both aspects on indoor thermal comfort. It can also contribute to future research concerning pollutant dispersion in the urban environment.

2. METHODOLOGY
Experiments performed by Kovar-Panskus et al. (2002) on a nominally 2D wind tunnel described airflow pattern inside a square street canyon (Width, Height = 285 mm) under the influence of buoyancy and wind perpendicular to the main canyon axis. The importance of the buoyancy effect inside street canyons can be characterized by the Froude (Fr) number which is defined as \( Fr = U_{ref}^2 / (gH(T_w - T_{ref})/T_{ref}) \), where \( U_{ref} \) is the free-stream reference wind speed above the top of the canyon (m s\(^{-1}\)), \( g \) the acceleration due to gravity (m s\(^{-2}\)), \( H \) the height of the street canyon (m), \( T_w \) the heated wall temperature (K), \( T_{ref} \) the ambient reference temperature (K). The downstream street facade was heated to obtain a uniform temperature over its whole surface. The airflow pattern inside the street canyon was measured for four different Froude numbers: 2.03, 1.17, 0.73 and 0.27. The model scale configuration respond to an actual street canyon with \( W, H = 20 \text{ m} \), in which the heated wall has a temperature around 305 K (\( \Delta t = 5 \text{ K} \)) and a free-stream wind velocity of 3 m s\(^{-1}\). In this paper, the CFD simulations are first compared with the wind-tunnel experiments. Then, based on the experimental configuration at model scale, a combination of five values of reference wind speed (\( U_{ref} = 0.5 - 5 \text{ m s}^{-1} \)) and six values of temperature difference between the heated wall and the ambient air temperature (\( \Delta t = 10 - 93 \text{ K} \)) were analysed yielding 25 different Froude numbers, ranging from \( Fr = 0.27 \) to \( Fr = 268 \) as shown in Table 1.

<table>
<thead>
<tr>
<th>( \Delta t \text{ = 93 K} )</th>
<th>( U_{ref} = 0.5 \text{ m s}^{-1} )</th>
<th>1.0</th>
<th>2.0</th>
<th>3.0</th>
<th>5.0</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.27</td>
<td>1.17</td>
<td>4.61</td>
<td>10.37</td>
<td>28.8</td>
<td></td>
</tr>
<tr>
<td>0.5</td>
<td>2.03</td>
<td>8.08</td>
<td>18.2</td>
<td>50.5</td>
<td></td>
</tr>
<tr>
<td>0.77</td>
<td>3.07</td>
<td>12.27</td>
<td>27.6</td>
<td>76.7</td>
<td></td>
</tr>
<tr>
<td>1.34</td>
<td>5.37</td>
<td>21.47</td>
<td>48.3</td>
<td>134.2</td>
<td></td>
</tr>
<tr>
<td>2.68</td>
<td>10.73</td>
<td>42.94</td>
<td>96.6</td>
<td>268.3</td>
<td></td>
</tr>
</tbody>
</table>

\( \Delta t \) Corresponding author: Abel Tablada, Laboratory of Building Physics, Katholieke Universiteit Leuven, Kasteelpark Arenberg 40, 3001 Heverlee, Belgium. Fax: +32 16 321980, Tel: +32 16 321345, e-mail address: abel.tablada@bwk.kuleuven.be
2.1. Numerical model and boundary conditions

For the comparison between the CFD simulations and the wind-tunnel experiments the case with \( F_r = 2.03 \) is explained here because it has the largest differences. The canyon has an aspect ratio \( W/H = 1 \) (\( W, H = 285 \) mm). Since dimensions of the wind-tunnel canyon configuration are 70 times smaller than the actual canyon, the values of wind speed and wall temperature were adapted in order to keep the same Froude numbers. Therefore, for the model scale, the measured mean wind speed profile is a logarithmic law with a gradient wind speed \( U_{\text{ref}} = 1 \) m s\(^{-1}\), a \( T_{\text{ref}} = 300 \) K and the downstream wall heated up to 353 K. The aerodynamic roughness length \( y_0 = 1.6 \) mm, and a friction velocity \( u^* = 0.07 \) m s\(^{-1}\). A structured quadrilateral mesh of 7804 cells was used. The grid resolution was based on grid-sensitivity analysis. The commercial CFD code Fluent 6.1.22 (2006) was used to solve the 2D (RANS) equations and the continuity equation. Closure was obtained using the realizable k-\( \varepsilon \) model [Shih et al., 1995]. Pressure-velocity coupling is taken care of by the SIMPLE algorithm. The top of the computational domain was modelled as a slip wall (zero normal velocity and zero normal gradients of all variables). At the outlet, zero static pressure was specified. The thermal numerical simulations were first performed as steady state with a low Rayleigh number (e.g., \( 10^5 \)) (\( g = 0.0981 \) m s\(^{-2}\)). Next, unsteady simulations with first order discretisation schemes and the correct value \( g = 9.81 \) ms\(^{-2}\) are made. Finally, unsteady simulations with second order schemes are conducted to yield the final results. Body-force Weighted was used for pressure interpolation.

3. RESULTS

3.1. Comparison with wind-tunnel experiments

The airflow pattern for the case of \( F_r = 2.03 \) for both the experimental and the present study are illustrated in Figure 1 by means of velocity vectors in a cross-section of the cavity. The CFD simulation predicts the buoyancy effect on the main vortex structure, but the upward flow close to the downstream heated wall seems to be over-predicted which cause two counter-rotating vortices indicated in figure 1b as A: close to the heated wall and B: close to the ground on the left side. In the wind-tunnel experiment the upward flow close to the heated wall was not observed due to the position of the measurement points. However, the authors mention the possibility of that upflow but very close to the wall and not affecting considerably the structure of the main vortex. The two counter-rotating vortices on the CFD simulations (figure 1b) make the main vortex smaller with a centre C shifted to the right and the top of the cavity in comparison with the wind-tunnel experiments (figure 1a). The secondary vortex near the ground that in the experiments appeared for \( F_r < 1 \), is predicted by the CFD simulations for \( F_r = 2.03 \).

![Figure 1: Velocity vectors inside cavity (model scale) with aspect ratio \( W/H = 1 \). (a): wind tunnel experiment. (b): CFD simulations. The capital letters indicate: A, secondary vortex with a strong updrawn flow close to the heated wall; B, secondary vortex close to the ground on the left side; and C, centre of the main vortex. Contours of air temperature (°C) are also indicated. Note: The scale of arrows are not comparable between the graphs.](image1.png)

The differences on the airflow distribution inside the cavity provokes differences on the horizontal velocities. These differences are \( U/U_{\text{ref}} = 0.11 \) at the top and bottom of the cavity at \( X/W = 0.5 \) and \( U/U_{\text{ref}} = 0.07 \) close to the heated wall \( X/W = 0.92 \) around half of the height of the cavity. However, the predicted airflow distribution and velocities are very similar to the wind-tunnel experiment of Kovar-Panskus et al. in the case of \( F_r = 0.73 \), (not showed here) with \( U_{\text{ref}} =0.8 \) m s\(^{-1}\), a \( T_{\text{ref}} = 300 \) K (\( \approx 27\)°C) and the downstream wall heated-up to 393K (\( \approx 120\)° C).

3.2. CFD cases with 25 Froude numbers

Figure 2 illustrates velocity vectors and airflow distribution of 12 out of the 25 studied cases. The results show that for high Froude numbers (\( F_r > 30 \)), one single main vortex is formed, highlighting the predominant role of wind in
the airflow pattern in the street canyon. On the bottom corners of the canyon two very small vortices with a relative stagnant flow are also created. At lower Froude numbers (Fr < 30), more than one vortex are formed: wind and thermal stack-induced vortices. The thermal stack-induced vortex is formed along the heated facade of the street canyon, as expected. This vortex reaches the top of the street canyon by compressing the wind induced vortex to the opposite side of the canyon for Fr < 3. The transition from a single main vortex to the creation of a weaker secondary vortex near the ground (by unification of the two bottom corner vortices) occurs at Fr ≈ 6 - 7 while in the wind-tunnel experiments this transition is reported to occur at Fr ≈ 1.

Concerning the velocity magnitudes, their average at a distance of X/W = 0.16 from each of the two walls increases by decreasing the temperature difference between the heated wall and the ambient temperature for U_{ref} ≥ 1 m s^{-1}. Nevertheless, as expected, in the area very close to the heated wall, the vertical updrawn velocity increases when the temperature difference increases but only for U_{ref} ≤ 2 m s^{-1}.

4. DISCUSSION

The set of CFD simulations indicates that the thermal effect due to the heating of the wall is somehow over-predicted in comparison with the experiments. A secondary vortex near the ground appears at Fr ≈ 6 - 7 while in the wind-tunnel experiments this transition is reported to occur at Fr ≈ 1 (higher temperature difference and/or lower wind speed). Similar observation was made by Louka et al., (2001) in which real measurements on a street were compared with CFD simulations. The authors reported an over-estimation of the thermal effects on the flow within the street canyon which created two counter-rotating vortices and a secondary smaller eddy at the leeward side at the bottom of the street. The authors attributed the differences to the wall function for temperature used in the model as well as to the size of the grid cells close to the walls. They suggested a reformulation of the temperature wall conditions in terms of thermal fluxes based on the thermal balance of the walls and a 3D model in order to take into account lateral airflows which are common under low air speed. Similar airflow distribution

Figure 2: Velocity vectors and streamlines inside a square cavity (model scale) for a selected number of cases with different Froude numbers (12 out of 25). Note: The scale and density of arrows are not comparable between the graphs.
than the one obtained in this study with a large secondary vortex close to the heated wall and ground is also reported in the literature by authors who performed CFD simulations in street canyons [Sini et al., 1996; Xie et al., 2005 and Xie et al., 2007]. They have reported two counter-rotating vortices when the downstream wall is heated but no indication of the Froude numbers or canyon dimensions were reported which could help to identify a transition from single to multiple vortices. However, in the full-scale experiments performed by Niachou et al., (2007) on a street canyon under hot weather conditions, an ascending flow close to downstream warmer wall and a secondary vortex near the ground was observed when the asphalt street-air temperature differences in the central part of the street canyon increased ($6^\circ$C $\leq \Delta T \leq 14^\circ$C). These situations correspond with Froude numbers ranging from around 2.8 to 4.2, which are higher than the threshold of 1 reported by Kovar-Panskus et al.

Notwithstanding the possible slightly over-estimation of the thermal effect on the airflow pattern inside a square street canyon by CFD simulations, the main airflow pattern evolution in relation to the Froude number is predicted for this simple configuration with perpendicular wind flow in relation to the main canyon axis. However, the simulations presented in this paper were performed at model scale in order to be compared with the results from the experiments reported by Kovar-Panskus et al. At full scale with realistic street canyon dimensions and higher Reynolds number, results may be different. In addition, as noted by Li (2006), constant temperature on canyon walls is not a realistic situation. Therefore, a set of 3D CFD simulations is been performed by the authors at full scale ($W, H = 20$ m) with more realistic boundary conditions on the canyon walls and ground together with ray-tracing short and discrete ordinates long-wave radiation algorithms with different solar angles of incidence.

5. CONCLUSIONS

In this paper the relative importance of wind versus thermal stack on the airflow pattern in a street canyon is analyzed. This is done by using Computational Fluids Dynamics (CFD) for the Kovar-Panskus et al. experimental configuration with five values of wind speed and five values of temperature difference between a heated wall and the ambient air temperature which yields 25 different Froude numbers. The results show that for high Froude numbers ($F_r > 30$), one single main vortex is formed. From $F_r = 30$ till $F_r = 3$ a secondary stack-induced vortex develops along the heated wall. At lower Froude numbers, this secondary vortex reach the full height of the heated wall. The two bottom corner vortices unify creating a weak secondary vortex near the ground at $F_r = 6-7$ which is higher than the threshold of $F_r = 1$ reported by Kovar-Panskus et al. but closer to the $2.8 \leq F_r \leq 4.2$ from Niachou et al. at an actual street canyon. A possible over-estimation of the thermal effect by CFD calculations has also being noted in literature. Therefore, further work is being performed at full scale 3D CFD simulations in which short and long-wave radiation algorithms are included. Finally, the effect of external aluminium louver systems on the facades surface temperature and on the canyon airflow pattern will also be studied.

6. ACKNOWLEDGMENTS

This paper is part of a research project about external shading devices on building facades. The authors are grateful to Reynaers Aluminium NV for their financial support.

References