This article is based on a paper presented at the 40th AIVC - 8th TightVent & 6th Venticool Conference "From energy crisis to sustainable indoor climate" 15-16 October 2019, Ghent, Belgium

Influence of horizontal mounted flue gas exhaust systems on indoor air quality

Horizontal mounted flue gas exhaust systems are used to reduce the installation costs of combustion appliances. The position of the exhaust terminal must be carefully chosen to prevent smoke from entering the ventilation system of nearby buildings. In this paper, a method based on computational fluid dynamics is used to determine the suitable zones for the mounting of the exhaust terminal with respect to the ventilation air supply openings.

XAVIER KUBORN
Belgium Building Research Institute, Belgium

SÉBASTIEN PECCEU
Belgium Building Research Institute, Belgium

Keywords: indoor air quality, ventilation, horizontal mounted flue gas exhaust, smoke exhaust, pollutants dispersion around buildings

Context

Combustion appliances are used in many buildings to provide space heating and domestic hot water. These appliances emit smoke that mostly contains carbon dioxide and water vapour, but also, depending on the type of fuel and the quality of the combustion, unburned hydrocarbons such as carbon monoxide, soot, tars and particulate matter. These products must be kept away from the ventilation air supply openings to limit their impact on the indoor air quality (IAQ).

An efficient way to prevent the flue gas from entering the building is to place the exhaust terminal above the top of the roof (see **Figure 1**), as far as possible from the ventilation air supply openings. The wind velocity, combined with the buoyancy of the smoke, will move the plume away from the building and dilute it into the atmosphere.

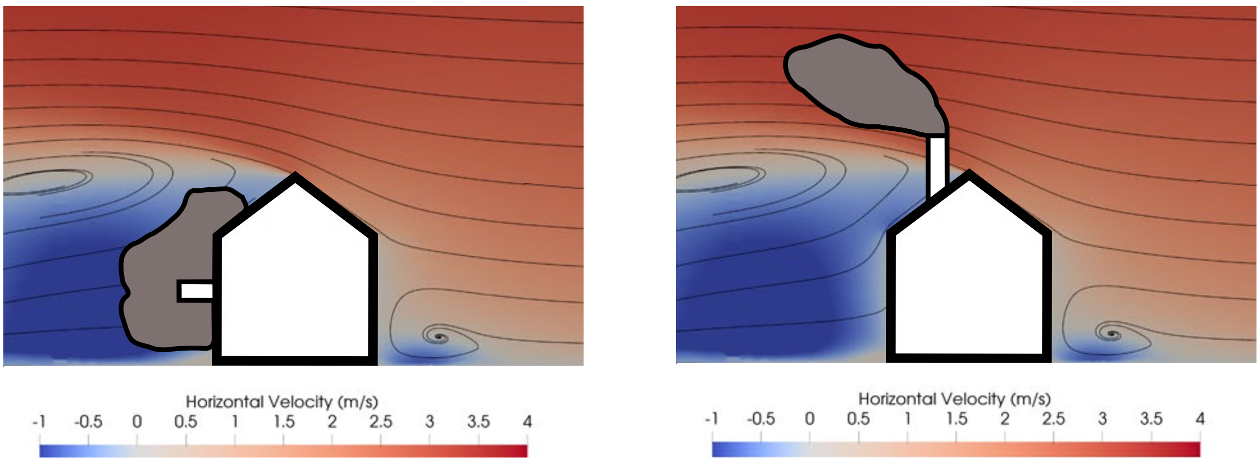


Figure 1. Streamlines of the wind flow around a building from a CFD simulation, with a superimposed qualitative representation of the smoke plume.

To reduce the installation costs of modern appliances, the chimney is often as short as possible and the exhaust terminal is mounted on a vertical wall, right next to the appliance. In that case, the plume might be partially trapped in a recirculation zone and remains close to the building with less dilution, increasing the risk of contamination. A minimal distance between the exhaust terminal and the ventilation air supply openings must be determined in order to avoid the recirculation of the pollutants inside the building.

Specific methods can be found in European and Belgian standards. A comparative example is given in **Table 1**. It highlights the discrepancies between the existing methods and the need for a tool to select the most appropriate one, or to develop a more widely accepted one.

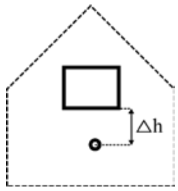
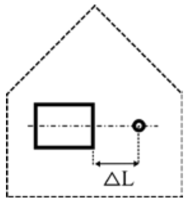
The minimal distance between the horizontal mounted flue gas exhaust system and the ventilation air supply openings depends on many parameters, including the heating power of the appliance, the temperature of the flue gas, the pressure of the exhaust and many others. But the most important parameter is the wind flow pattern around the building, that depends on the shape of the building and on the direction of the wind. Depending on the flow pattern and on the position of the exhaust terminal, the flue gas plume can be driven away from the building by the wind, or be taken back against it.

A relevant method to study the wind flow pattern and the flue gas dispersion around buildings is to perform computational fluid dynamics (CFD) simulations.

CFD and Numerical model

The complex physics of the problem is described by a set of equations (the Navier-Stokes equations) that are solved by a CFD tool. The concentration field of the pollutants around the building is determined from the solution of the equations. For industrial (non-academic) research, an alternate version of the Navier-Stokes equations (RANS equations: Reynolds-Averaged Navier-Stokes equations) and a turbulence model ($k - \omega$ SST) are used, as it has been proven appropriate in other studies dealing with flow around buildings (Ramponi R, 2012). The numerical simulations are carried out on a four facades isolated building located on a flat ground. By adjusting the geometrical parameters of the numerical model, the geometry can be extended to that of multi-storey buildings and terraced houses, as represented in **Figure 2** on the following page. The exhaust flow, temperature, and position as well as the wind velocity and direction are also adjustable parameters. For each set of parameters, a steady-state solution is computed.

Table 1. *Discrepancies between the different standard methods.*

Geometrical configuration	Standard reference	Recommended distance
	NBN EN 15287-2(2008) NBN B 61-002(2006) NBN D 51-003(2014)	Δh = 30 cm Δh = 320 cm Δh = 500 cm
	NBN EN 15287-2(2008) NBN B 61-002(2006) NBN D 51-003(2014)	ΔL = 30 cm ΔL = 340 cm ΔL = 100 cm

Pollutant field on a façade

To visualize the computed pollutant field near the facades of building, an iso-contour at a dilution of 100 is shown in **Figure 3** on the following page. This specific iso-contour represents the locations in space where the concentration of the pollutant is 100 times lower than that at the exhaust terminal. A dilution of 100, for gas-fired appliances, is considered to be representative of a sufficient air quality to be used for building ventilation

Figure 3 also shows that the smoke plume goes backward against the wind and along the façade as the exhaust terminal is located in a recirculation zone. The iso-contour representing a dilution of 100 is not connected to the exhaust terminal, as either the initial velocity of the flue-gas or the wind pattern initially moves it away from the building before pulling it back against the façade.

Yearly analysis and similarities

The result presented in **Figure 3** is an instant picture of a specific set of operating parameters, but it does not reflect the risk encountered in real life, as the many parameters are dependent on the environmental conditions, including the wind velocity, the wind direction and the outside temperature, that are variable in essence. If a yearly overall effect is to be accounted for, a statistical approach using all the relevant environmental parameters needs to be used. However, this approach implies that many different numerical simulations need to be done

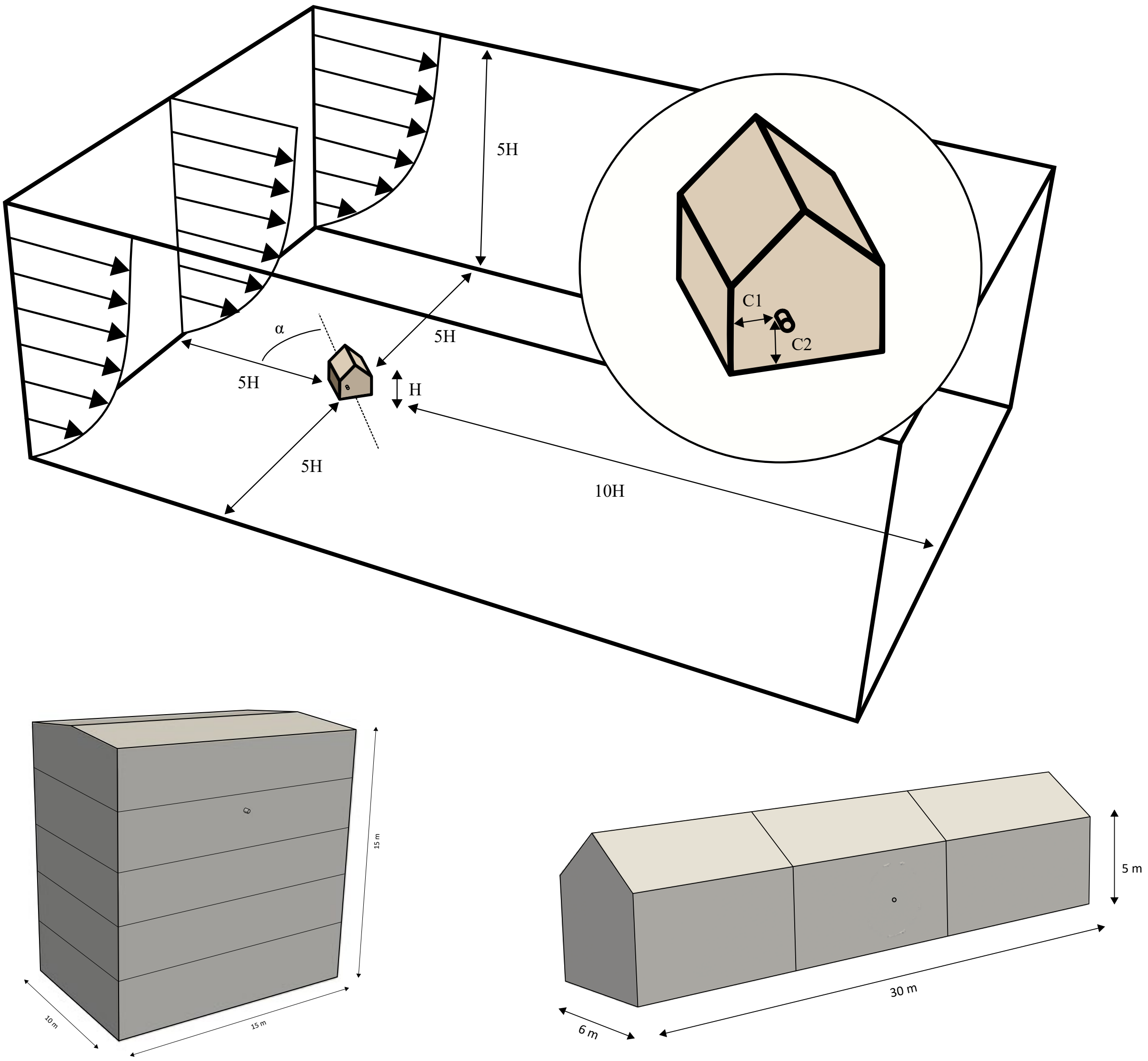


Figure 2. The numerical domain (above) and examples of geometries (below) derived from a four facades building.

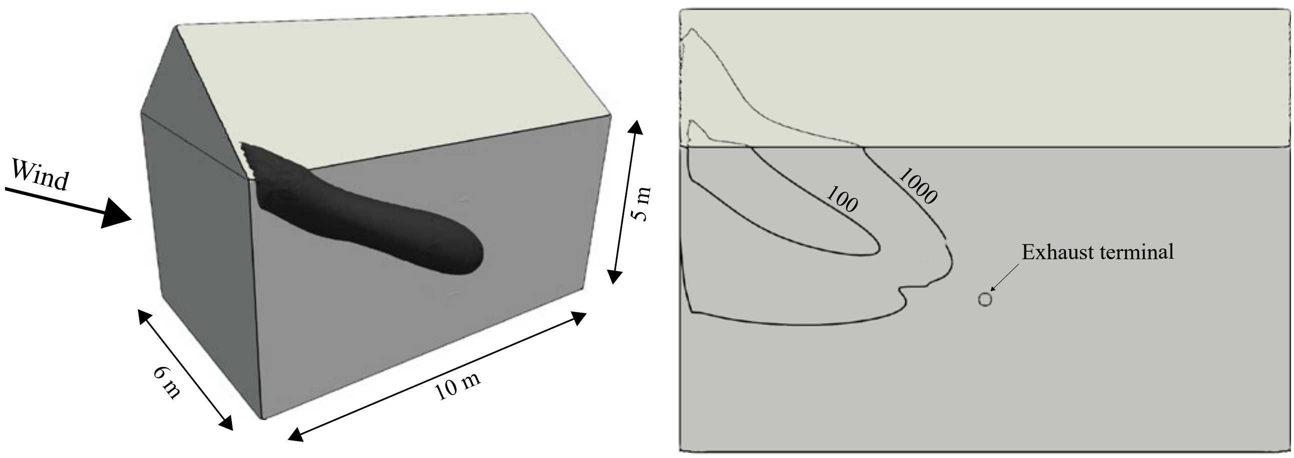


Figure 3. Representation of the iso-contours of concentration of the flue gas around a detached house.

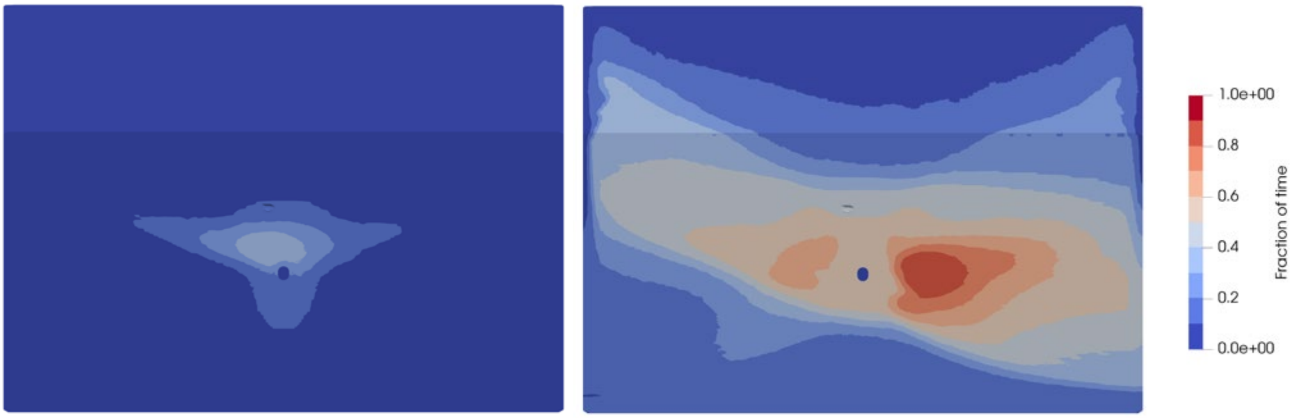


Figure 4. Probability that the dilution of a pollutant is below a threshold of 100 or 1000 on a yearly basis.

in order to get all the relevant results. Fortunately, many similarities can be identified, and the results of several numerical simulations can be used to complement other similar cases, using an appropriate scaling. For instance, the flow pattern around the building remains the same for any wind velocities above 2 m/s, which reduces the need to perform new numerical simulations for different wind velocities. However, the wind velocity is accounted for in the scaling as it increases the dilution of the smoke plume.

Assuming that the heating appliance is operating at nominal power, only height numerical simulations (one per main wind direction) are needed to determine the impact of the smoke on the façades of buildings. Knowing the wind velocity and wind direction for each hour of the year, these height numerical simulations and the scaling procedure are used to compute the pollutant field for each hour, which in turn is used to compute the probability that a concentration threshold is reached or exceeded on the façades of the building. Such probability is shown in **Figure 4** for a dilution of 100 (left-hand side of the figure) and 1000 (right-hand side of the figure).

Future work

The next steps of this project are to further validate the method and to develop a tool, based on these numerical results, that could be used by the heating specialist to determine the suitable locations for the exhaust terminal for many environmental parameters and building geometries. ■

Acknowledgments

This paper is written in the frame of two pre-normative studies called In-Vent-Out and In-Vent-Out 2, funded by the Belgian Federal Public Service and the Belgian Building Research Institute.

This article is based on a paper presented at the 40th AIVC - 8th TightVent & 6th venticool Conference, 2010 "From energy crisis to sustainable indoor climate - 40 years of AIVC" held on 15-16 October 2019 in Ghent, Belgium.

References

Carlo Gualtieri, A. A. (2017). On the Values for the Turbulent Schmidt Number in Environmental Flows. *Fluids*.

Fougairolle, P. (2009). Caractérisation expérimentale thermo-aéraulique d'un jet transverse impactant ou non, en turbulence de conduite.

Gousseau P, B. B. (2010). CFD simulation of near-field pollutant dispersion on a high-resolution grid: a case study by LES and RANS for a building group in downtown Montreal. . *Atmospheric Environment 45*, 428-438.

Kiyoshi Uehara, S. W. (2003). Studies On Critical Reynolds Number Indices Forwind-Tunnel Experiments On Flow Within Urban Areas. *Boundary-Layer Meteorology 107*, 353-370.

Ramponi R, B. B. (2012). CFD simulation of cross-ventilation for a generic isolated building: impact of computational parameters. *Building and Environment 53*, 34-48.

Tominaga, Y. M. (2008). AIJ guidelines for practical applications of CFD to pedestrian wind environment around buildings. *Journal of wind engineering and industrial aerodynamics*, 1749-1761.