A study of the influence of the position of a chimney terminal on the vertical walls of a building on the air quality of the ventilation air supply

Xavier Kuborn[1], Sébastien Pecceu[1]

1 Belgium Building Research Institute
Avenue Pierre Holoffe, 21
1342 Limelette, Belgium

ABSTRACT

Combustion appliances are used in many buildings to provide space heating and domestic hot water. These appliances emit smoke that contains pollutants that must be kept away from the ventilation air supply of the building, to limit their impact on the indoor air quality (IAQ). An efficient way to prevent those pollutants from entering the ventilation circuit is to place the chimney terminal above the top of the roof, as far as possible from the air supply openings. However, to reduce the installation costs, the chimney is often as short as possible and the terminal is mounted in a vertical wall, right next to the appliance. In this case, a minimal distance between the terminal and the ventilation openings must be determined.

Methods to determine such distances can be found in several European and national standards. But most of these methods are overly simplistic and do not reflect the complexity of the physical phenomena at stake, namely the wind flow pattern around the building. The aim of this paper is to present a method, based on computational fluid dynamics simulations, that is able to determine the zones on the vertical walls of a building, with respect to the position of the chimney terminal, where the concentration of the pollutants is sufficiently low so that the air can be used for ventilation.

Preliminary results show that these zones are strongly dependant on the shape of the building and on the wind direction with respect to the building. A yearly analysis of the effect of the changing wind direction on the flue gas plume on the vertical walls of the building is also presented.

KEYWORDS

Indoor air quality, ventilation, chimney terminal, pollutants dispersion

1 INTRODUCTION

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 are pollutants that must be kept away from the ventilation air supply of the building, to limit their impact on the indoor air quality (IAQ). Combustion appliances include, but are not limited to, gas boilers, oil boilers, wood stoves, pellets stoves and open fires. The amount of flue gas that is emitted by the appliance depends on its heating power and on the operating duration.

An efficient way to prevent the flue gas from entering the building is to place the chimney terminal above the top of the roof (see Figure 1), as far as possible from the 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. However, to reduce the installation costs of modern appliances, the chimney is often as short as possible and the terminal is mounted on a vertical wall, right next to the appliance (see Figure 1). In that case, the plume might be partially trapped in a recirculation zone and remain close to the building with less dilution, increasing the risk of contamination. A minimal distance between the terminal and the ventilation openings must be determined in order to avoid the recirculation of the pollutants inside the building. Methods to determine this distance can be found in European and Belgian standards. A comparative example between three methods is given in chapter 2. 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.

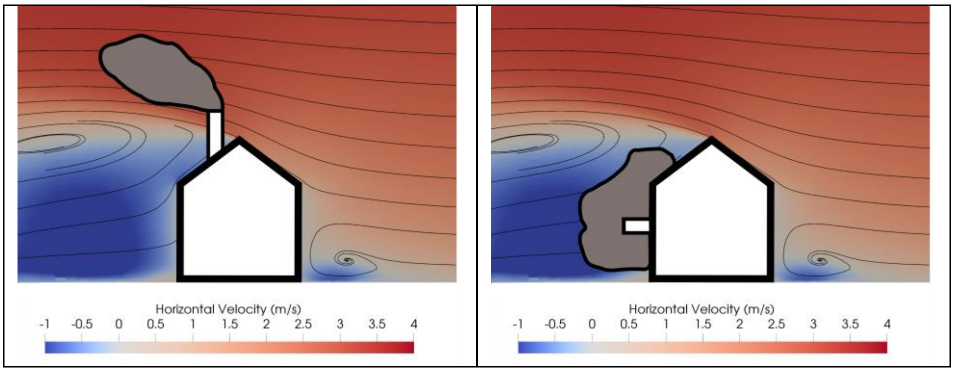


Figure 1 : streamlines of the wind flow around a building from a CFD simulation, with a superimposed qualitative representation of the smoke plume with respect to the position of the terminal.

Estimating a minimal distance between the chimney terminal and the ventilation openings is not an easy task, as it 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 parameters that have the strongest influence on the plume dispersion are the wind turbulence and the wind flow pattern around the building. The wind flow pattern depends on the shape of the building and on the direction of the wind. Therefore, the flue gas plume can be driven away from the building by the wind, or be taken back against it.

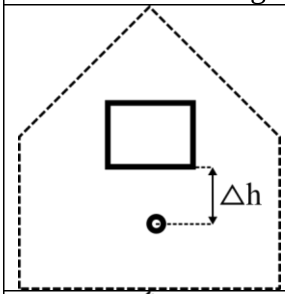
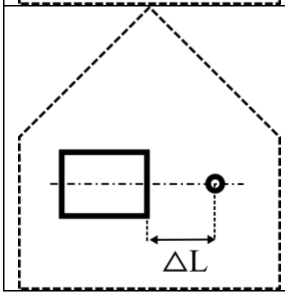
A relevant method to study the wind flow pattern and the flue gas dispersion around buildings is to perform computational fluid dynamics (CFD) simulations. The physical phenomena are described by a mathematical model that consists of a set of equations and phenomenological coefficients. CFD is used to solve the set of equations and to provide the relevant physical fields, such as the flue gas concentration. Although wind driven pollutant dispersion around buildings is a complex topic, it has largely been studied in the literature with the use of CFD. The mathematical model and the numerical method are described in chapter **Error! Reference source not found.** while preliminary results are shown in chapter 4.

While the subject of this paper is to describe a method to prevent a negative impact of the position of the chimney terminal on the IAQ, the results are to some extent also useful for the positioning of ventilation exhausts and kitchen hoods.

2 STANDARD METHODS

Several European and Belgian standards provide methods for positioning the chimney terminal with respect to the ventilation air supply openings. But most of these methods are overly simplistic and do not reflect the complexity of the physical phenomena at stake, namely the wind flow pattern around the building. Additionally, many discrepancies can be found between the methods. For instance, in Belgium, the minimal distance between the chimney terminal of a 30 kW room-sealed gas appliance, located on a vertical wall and a ventilation opening located on the same wall, can be determined by the following standards : NBN EN 15287-2(2008), NBN D 51-003(2014) or NBN B 61-002(2006). For each standard, the minimal distance for two geometrical configurations is shown in Table 1.

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)	Δh = 30 cm Δh = 340 cm Δh = 100 cm

There are obviously strong discrepancies between the values given by the different standards, with certain methods being more conservative than others. But the most conservative method is not always the same, depending on the selected configuration. Regardless of the quality of the given answer, the discrepancies between the standards show that there is a need for a better understanding of the problem and for a more relevant and widely accepted method for solving it.

There is no explanation associated with the methods described in the standards NBN EN 15287-2 and NBN D 51-003. However, the method given in NBN B 61-002(2006), called the method of the dilution factor, consists of a mathematical formula that leaves room for interpretation. Using this formula, a dilution of 100 of the flue gas (for gas appliances) must be obtained to be able to use the air for the ventilation of the building. The dilution is proportional to the height difference and the distance between the terminal and the ventilation opening and inversely proportional to the power of the appliance, which makes sense from a physics standpoint.

Although it is not possible, based only on the comparison between the methods, to determine which one is the most accurate, the value of a minimal dilution of 100 will be used as a reference during the development of a new method. This reference will allow for comparisons between the existing methods and the new one, regardless of the relevance of the value of 100 itself.

3 PARAMETRIC CHAIN

The CFD simulations presented in this paper have been carried out using OpenFOAM 2.4.0. OpenFOAM is a free and open source CFD software. Based on OpenFOAM, a parametric chain has been specifically developed to simulate the dispersion of pollutants around four facades isolated buildings on flat ground. The different elements of the chain are described below.

3.1 Mathematical model

As mentioned earlier in this paper, the dispersion of pollutants in the vicinity of a building is mostly driven by the wind flow pattern around the building, by the turbulence of the flow and to a lesser extent by the buoyancy of the flue gas. Buoyancy results from a temperature-induced density difference between the flue gas and the atmosphere. Hence, the energy equation needs to be solved to account for this phenomenon. Accordingly, the Navier-Stokes equations, i.e. the conservation equations of mass, momentum and energy need to be solved to capture the physics of the problem. The Navier-Stokes equations are not recalled here, since they are already largely reported in the literature.

Turbulence modelling

For industrial research, as is it the case for this study, turbulence is not directly solved using the Navier-Stokes equations, as it would require too much computational power. Turbulence is instead modeled, using an alternate version of the Navier-Stokes equations (RANS equations : Reynolds-Averaged Navier-Stokes equations) and two closure equations to model the turbulence. In this study, the $k - \omega\ SST$ closure model is used as it has been proven appropriate in other studies dealing with flow around buildings. (Ramponi R, 2012). These equations are also not recalled here for the same reason.

Atmosphere stratification and wind profile

The stability of the atmosphere describes its tendency to discourage or encourage the vertical motion of a parcel of air. Atmospheric stability is an important parameter when the dispersion of pollutants is studied on larger scales, e.g. the pollution from an industrial chimney or from a chemical accident. However, when the study is limited to the vicinity of a building, such phenomenon is generally neglected. In the present analysis, a neutral atmosphere, that do not encourage nor discourage the vertical motion of a parcel of air, is considered. The only resulting vertical motions are generated by either the wind flow pattern induced by the building or by the buoyancy of the flue gas. A logarithmic wind profile is used for the inlet boundary condition of the wind velocity field. The logarithmic profile gives the vertical distribution of the wind velocity and takes into account the terrain roughness.

Heat and species convection/diffusion

The density of the flue gas, that is responsible for its vertical motion associated with buoyancy, is estimated using an ideal gas law.

The flue gas is considered to be composed of a single fictive species whose concentration is equal to one at the chimney terminal and that has the same properties as air. An additional species equation (1) is added to the set of the Navier-Stokes equations. Solving this equation gives the concentration field of the pollutant in the computational domain.

$$\frac{\partial C}{\partial t} + \mathbf{v} \cdot \nabla C - \frac{\mu}{\rho \mathrm{Sc}} \nabla^2 \mathrm{C} = 0\,, \tag{1}$$

where C is the species concentration, $\mathbf{v}$ is the velocity field tensor, μ is the is dynamic viscosity of air, ρ is the density of air and Sc is the dimensionless Schmidt number. Using RANS methods, a turbulent viscosity is estimated by the closure equations. A turbulent Schmidt number is the ratio between turbulent momentum diffusion and turbulent mass diffusion. A wide range of turbulent Schmidt number values (from 0.3 to 1.3) are reported in the literature depending on the configuration (Carlo Gualtieri, 2017) (Gousseau P, 2010); a default intermediate value of 0.7 is used here.

3.2 Geometry of the problem, domain size and mesh generation

The mathematical model needs to be constrained in order to capture the physics of the problem. The studied geometry is a four facades isolated building with a two-sided roof (see Figure 2). The geometrical parameters are the length, the width and the height (of the cornice) of the building as well its roof angle. Combining these parameters, the parametric chain is able to generate many different building shapes, including, but not limited to, detached, semi-detached or terraced homes and apartments buildings. Additionally, the roof angle can be set to zero, which allows to generate flat roofs. The characteristics of the chimney are defined by four additional parameters: its diameter, its length and the two relative coordinates of its position on any vertical façade.

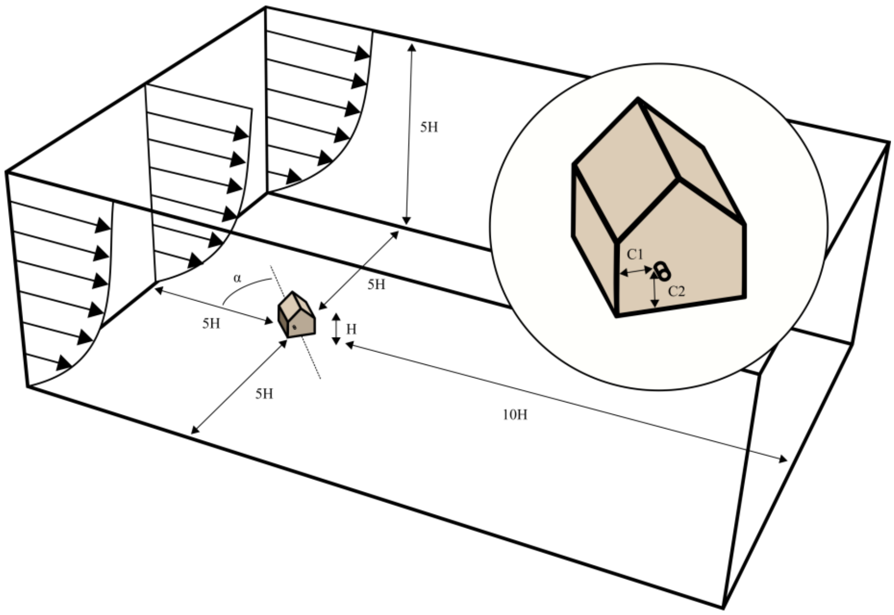


Figure 2 : representation of the computational domain with its minimal dimensions with respect to the height of the building and the logarithmic wind profile at the inbound condition.

The dimensions of the computational domain, as shown in Figure 2, are sufficiently large (Tominaga, 2008) to avoid an impact of the boundary conditions on the final results in the vicinity of the building.

The computational mesh is generated using the OpenFOAM built-in tool, *snappyHexMesh*. It is a locally refined mesh, that uses a method where space is filled by cubes or hexahedra elements that are successively refined by a factor two from the largest to the smallest size. The mesh sizes go from 2.5 m (coarsest size, far from the building) to 2.0 cm (at the chimney terminal).

3.3 Other operating parameters

Three operating parameters are needed to describe the atmospheric conditions: wind velocity, wind direction (relative to the orientation of the building) and outside temperature. The wind velocity is given by the logarithmic profile at the domain inlet while the wind direction is given by choosing the orientation of the building with respect to the inlet condition (see Figure 2). The outside temperature is uniform inside the computational domain at the beginning of the simulation. Finally, two operating parameters are needed to describe the flue gas: its flow and its temperature. The flow mirrors the power of the connected appliance. Associated with the diameter of the chimney, it determines the exhaust velocity. During a simulation, the appliance operates at nominal power, hence the flow of the flue gas remains constant.

3.4 Assessment of the method

Although RANS is the best affordable method for doing CFD in industrial applications, it overestimates the recirculation zone in the wake of the building. Therefore, more pollutants might be redirected towards the building than in real life. Additionally, the OpenFOAM solver used in this study gives a steady state solution of the problem. In steady state, the dispersion of the pollutant is underestimated as there is no sudden change in the wind direction that will inevitably break the flue gas plume apart. Also, the neutral atmosphere hypothesis, where the only vertical motions are induced by the buoyancy or the wind flow pattern, will also underestimate the dispersion as it decreases the vertical mixing of the flue gas plume. Finally, the studied geometry is that of an isolated building on flat ground, with no obstacles that may change the wind flow upstream of the building and consequently the wind flow pattern around the building.

4 PRELIMINARY RESULTS

For a set of geometrical and operating parameters, the parametric chain described in chapter 3 is able to solve the steady-state dispersion of the flue gas and to compute its concentration field in the whole computational domain. A first example is given in Figure 3 for a single-family detached house. The chimney terminal is located at the center of the front façade. The left-hand side of Figure 3 shows a 3D iso-contour of concentration representing a dilution of 100 with respect to the concentration at the exhaust. The right-hand side shows a 2D representation of two iso-contours representing dilutions of 100 and 1000 (concentrations of 0.01 and 0.001 respectively). A dilution of 100 is considered to be representative of a sufficient air quality to be used for building ventilation, as written in chapter 2.

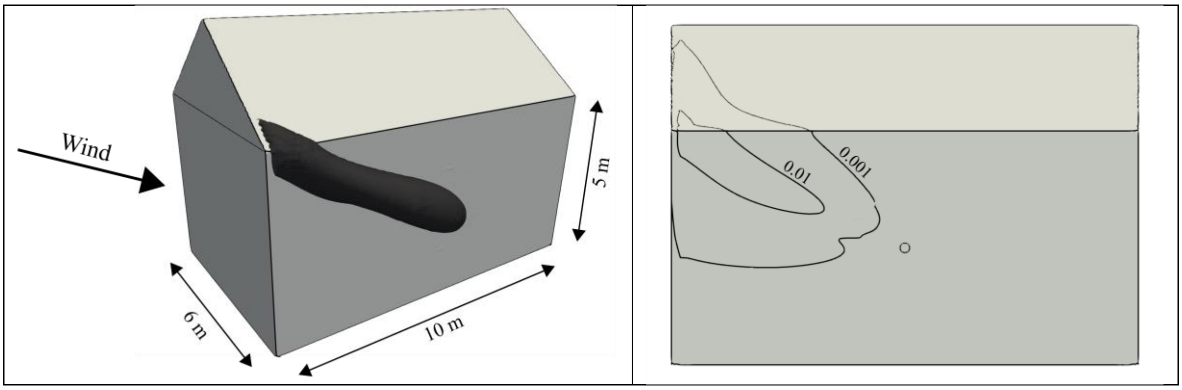


Figure 3 : representation of the iso-contours of concentration of the flue gas around a detached house.

Figure 3 also shows that the flue-gas plume goes backward against the wind and along the façade as the chimney terminal is located in a recirculation zone. The iso-contour representing a dilution of 100 is not connected to the chimney 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.

A second similar example is shown in Figure 4 for a five dwellings apartments building and very low wind velocity (0.5 m/s). The chimney terminal is located on the second floor of the building. The flue-gas plume elevates mostly vertically, as the wind velocity is low, but it also goes backward with respect to the wind flow as the terminal is located in a recirculation zone. The length of the iso-contour representing a dilution of 100 spans across three dwellings.

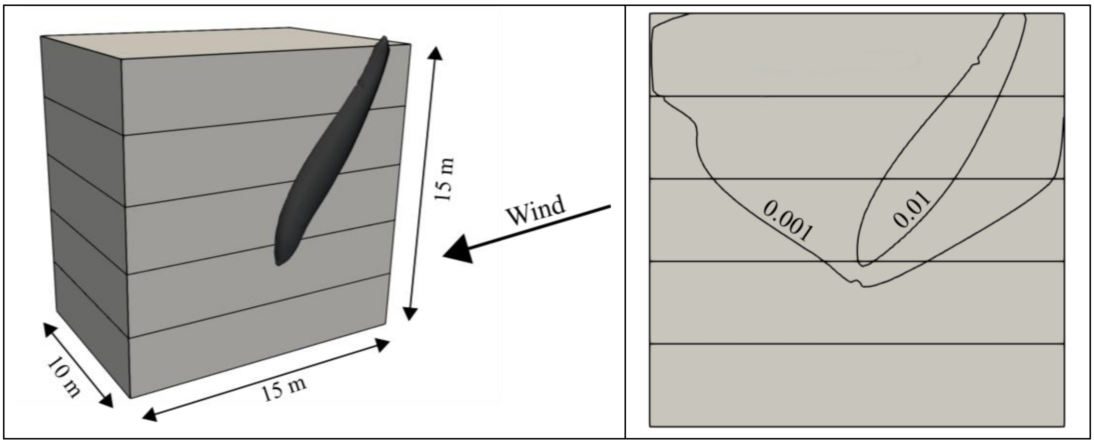


Figure 4: representation of the iso-contours of concentration of the flue gas around an apartments building.

A third example, presented in Figure 5, features the same operating parameters as the example shown in Figure 4, with the exception that the chimney terminal is located in the last dwelling. The flue-gas plume has a totally different shape that can be explained by the wind flow pattern around the building.

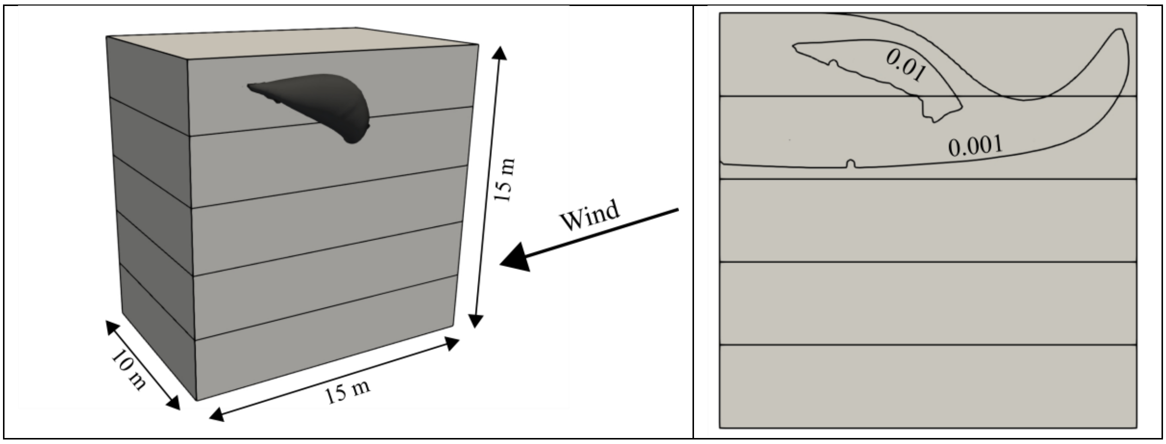


Figure 5: representation of the iso-contours of concentration of the flue gas around an apartments building

These results are instant pictures of a specific set of operating parameters, but they do not reflect the risk encountered in real life, as the many parameters are dependent on the meteorological conditions, that vary in time and that have a certain probability to occur. Such meteorological analysis is discussed in chapter 4.2, while a method to reduce the number of simulations needed to achieve such analysis is discussed in chapter 4.1.

4.1 Similarities and simplifications

The physics of the dispersion of the flue gas around a building involves many parameters that have been described in chapter 3. The geometrical parameters represent a specific building configuration while the operating parameters describe the environment. For a specific building configuration, a single numerical simulation using a single set of environmental parameters is insufficient to determine the overall potential influence of the flue gas on the IAQ. Indeed, the environmental parameters, including the wind velocity, the wind direction and the outside temperature, are variable in essence. They vary in time on a minute based scale but also on a daily scale with the night and day cycles as well as on a yearly scale with the different seasons.

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 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.

The Reynolds number is one of the most used dimensionless number in the field of fluid dynamics. For an incompressible and isothermal flow, two similar geometries (same shape but different scales) with the same Reynolds number will have identical flow fields. The case of the wind flow around a building is more complex, but the literature shows that the flow field remains similar for a wide range of Reynolds numbers (Kiyoshi Uehara, 2003). The Reynolds number takes into account the fluid characteristics (viscosity, density) and velocity as well as a characteristic dimension of the flow (the size of the building). It means that for a similar building geometry, the wind velocity has no influence on the flow pattern around the building.

Another characteristic of the problem is the jet generated by the exhaust of the flue gas in the main wind flow, that is often referred to in the literature as the “jet in crossflow” problem. This complex problem is approximated by the ratio of the momentum of the two flows (Fougairolle, 2009) :

$$J = \frac{\rho_j U_j^2}{\rho_\infty U_\infty^2} \tag{2}$$

If densities of both fluids are identical, it further simplifies to a velocity ratio U_j/U_∞. When the flue gas velocity is much lower than the wind flow velocity, this problem is similar to that of a local pollutant source, that can be modeled by the gaussian dispersion model. It is a common model for pollutant dispersion, where the concentration field of the pollutant is proportional to the pollutant source and inversely proportional to the wind velocity.

Empirical assessment

Considering this reasoning, the flow pattern around a building should be similar for any wind velocity and the pollutant concentration should be proportional to the flue gas flow (the source), and inversely proportional to the mainstream wind velocity. For a given building geometry (dimensions and chimney location), only one numerical simulation per wind direction should be able to represent the pollutant dispersion for all other operating parameters (wind velocity, outside temperature, flue gas flow, outside temperature and flue gas temperature).

This reasoning is not perfect, since the pollutant source has a non-null velocity at the chimney outlet and the temperature difference between the flue gas flow and the atmosphere should have a more significant impact when the Richardson number is high. The Richardson number is the ratio of the buoyancy to the flow shear.

In order to assess this reasoning, several numerical simulations have been carried out with a single geometrical configuration and varying wind velocities (V) and flue gas flow (Q), all other operating parameters remaining constant. The pollutant concentration fields (S) are then compared using the following scaling:

$$S_{scaled} = S_{sim} \frac{Q_{ref}}{Q_{sim}} \frac{V_{sim}}{V_{ref}} \tag{3}$$

The ‘ref’ and ‘sim’ suffixes referring respectively to a reference case and cases with other operating parameters. For all cases, the flue gas temperature is 100°C and the atmosphere temperature is 0°C. If the reasoning makes sense, all scaled concentration fields should be equal. The reference case is presented in Figure 6, while the scaled concentration field are presented in Figure 7. The results are shown on a horizontal plane whose height is the same as the chimney axis.

Reference case: Q_{fumes} =100 m³/h ; V_{wind} = 3 m/s

Figure 6 : iso-contour of the concentration field (log scale) for the reference case.

Three out of the four scaled simulations give nearly identical concentration fields, with the case with low wind velocity remaining qualitatively close. More numerical simulations with an increased range of the operating parameters should indeed be necessary to better assess the reasoning, but it already looks promising.

Low source case: $Q_{flue\ gas}$ =50 m³/h ; V_{wind} = 3 m/s

High source case: $Q_{flue\ gas}$ =200 m³/h ; V_{wind} = 3 m/s

Low wind case: $Q_{flue\ gas}$ =100 m³/h ; V_{wind} = 1 m/s

Strong wind case: $Q_{flue\ gas}$ =100 m³/h ; V_{wind} = 3 m/s

Figure 7 : scaled concentration fields (log scale) for numerical simulations with identical geometries but different flue gas flow and wind velocities.

4.2 Principle of the yearly analysis

Assuming that the heating appliance (or the ventilation installation) is always operating at nominal power, i.e. that the flue gas flow and temperature remain constant, the pollutant field only depends on the wind velocity and wind direction. In the example shown in this paper, a given building configuration (dimensions, orientation and chimney terminal position) is associated with a wind file representative of Brussel’s climate (see Figure 8). The corresponding hourly wind velocities and directions are decomposed into height major directions, that will be simulated using the parametric chain.

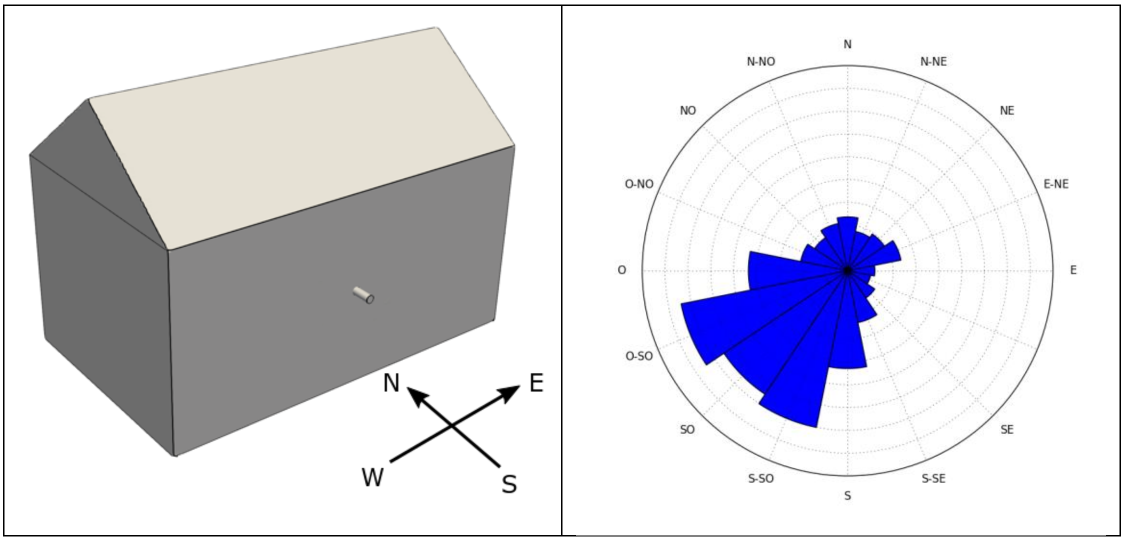


Figure 8: building configuration and occurrence probability for wind directions in Brussel.

Using these height numerical simulations and the scaling procedure described in chapter 4.1, the pollutant field can be determined for each hour of the year, based on the corresponding wind direction and velocity. On a yearly basis, the probability that a concentration threshold is reached or exceeded on the façades of the building can be determined. Such probability is shown in Figure 9 for a dilution of 100 (left-hand side of the figure) and 1000 (right-hand side of the picture). A dilution of 100 corresponds to a pollutant concentration of 1/100, assuming that the pollutant concentration is equal to 1 at the terminal.

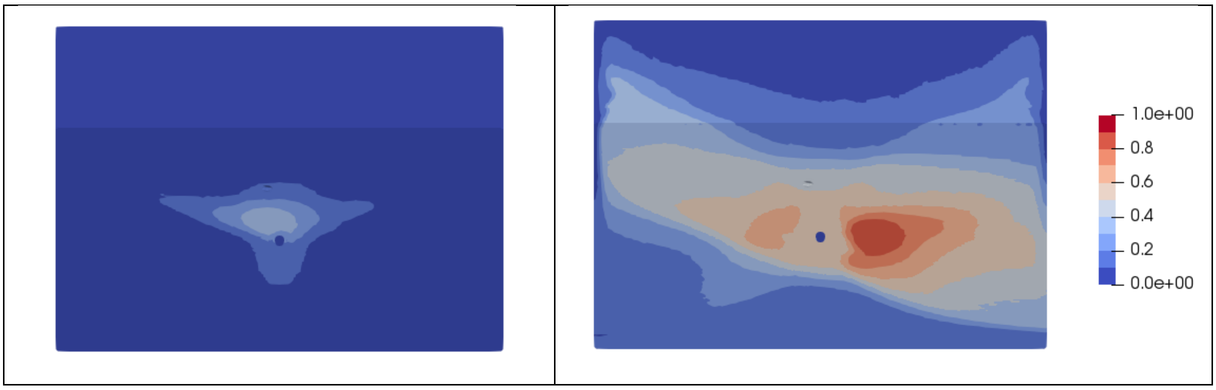


Figure 9 : probability that the pollutant concentration exceeds a threshold of 1/100 or 1/1000 on a yearly basis.

Depending on the value of the dilution coefficient that is considered to be acceptable, the appropriate "map" of the façades of a building could be used to determine where the ventilation openings could be located. The determination of the acceptable dilution coefficient and the acceptable probability that such dilution is not reached falls out of the scope of this paper.

5 FUTURE WORK AND CONCLUSIONS

This paper describes a method for the positioning of the chimney terminal with respect to the ventilation openings on the façades of buildings. The described method provides "maps" of the façades that show the probability that the flue gas is sufficiently diluted into the atmosphere to use the air for the ventilation of the building. When refined and validated, such maps could be used in standards as methods for the positioning of chimney terminals and ventilation openings.

Many improvements could be done to this method. Specifically, the appliance could operate at powers that are below nominal power, according to real-life operating patterns. That would reduce the amount of emitted pollutant and potentially increase the size of the zones where the ventilation openings could be placed. Also, the scaling procedure could be further validated for an increased range of operating parameters, for instance when the wind velocity is lower, when the exhaust velocity is higher and when the buoyancy is more important. When validated maps are obtained, the results could also by confirmed be doing on-site monitoring nearby chimney terminals located on façades. Finally, summarizing all the available results into a method that is simple enough to be used in standard as rules of good practice could be the most difficult challenge of them all.

6 ACKNOWLEDGEMENTS

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.

7 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.