Received 22 December 2019
Revised 4 March 2020
29 July 2020
15 September 2020
Accepted 17 September 2020

Scaling procedure for designing accidental gas release experiment

Alberto Moscatello and Anna Chiara Uggenti
Department of Energy, Politecnico di Torino, Torino, Italy

Gaetano Iuso, Domenic D'Ambrosio and Gioacchino Cafiero
Department of Mechanical and Aerospace Engineering, Politecnico di Torino, Torino, Italy, and

Raffaella Gerboni and Andrea Carpignano
Department of Energy, Politecnico di Torino, Torino, Italy

Abstract

Purpose – The purpose of this paper is to present a procedure to design an experimental setup meant to validate an innovative approach for simulating, via computational fluid dynamics, a high-pressure gas release from a rupture (e.g. on an offshore oil and gas platform). The design is based on a series of scaling exercises, some of which are anything but trivial.

Design/methodology/approach – The experimental setup is composed of a wind tunnel, the instrumented scaled (1:10) mock-up of an offshore platform and a gas release system. A correct scaling approach is necessary to define the reference speed in the wind tunnel and the conditions of the gas release to maintain similarity with respect to the real-size phenomena. The scaling of the wind velocity and the scaling of the gas release were inspired by the approach proposed by Hall *et al.* (1997): a dimensionless group was chosen to link release parameters, wind velocity and geometric scaling factor.

Findings – The theoretical scaling approaches for each different part of the setup were applied to the design of the experiment and some criticalities were identified, such as the existence of a set of case studies with some release parameters laying outside the applicability range of the developed scaling methodology, which will be further discussed.

Originality/value – The resulting procedure is one of a kind because it involves a multi-scaling approach because of the different aspects of the design. Literature supports for the different scaling theories but, to the best of the authors' knowledge, fails to provide an integrated approach that considers the combined effects of scaling.

Keywords Risk assessment, Scaling, Multi-scale, Wind tunnel, Computational fluid dynamics, Experimental setup, Pressurised gas release

Paper type Research paper

1. Introduction

The Deepwater Horizon accident in the Gulf of Mexico is dated back to year 2010. Since then, many initiatives have been carried out to prevent similar or even less impacting events on

The research presented in this paper has been sponsored by the Italian Ministry of Economic Development's Directorate General for Safety – National Mining Office for Hydrocarbons and Georesources.

offshore oil and gas installations. Among these, the European Union has issued a specific directive (30/2013/EU) that requires operators of offshore installations to prepare a report on major hazards to be submitted for approval to the local (national) competent authorities. Reports on major hazards build on the production of extensive risk assessments and a cornerstone of these analyses is the evaluation of consequences because of failures in the systems.

Oil and gas offshore installations are complex structures composed of decks (floors) containing piping and process units, usually dealing with flammable and toxic fluids operating under pressure. One of their critical characteristics is the congestion of spaces where operators work and where accidental releases may take place and evolve.

To simulate the evolution of a high-pressure gas release in such a congested space since the design phase (but also afterwards, to identify hazardous scenarios to prevent), computational fluid dynamics (CFD) has proven suitable and it is now frequently adopted by operators and designers.

The SEADOG DENERG laboratory in the framework of a large project funded by the Italian Ministry of Economic Development, in support of safety on-board oil and gas platforms, has proposed a new approach (Carpignano *et al.*, 2017), presently under development, which aims at reducing the computational time and improving the accuracy of simulation results obtained with parametric models.

The approach is based on splitting the high-pressure gas release in two phases:

(1) the initial rupture in a pipe or in a vessel that produces a highly under-expanded, supersonic jet; and

(2) the subsequent low-speed dispersion in the rest of the platform.

The magnitude of the physical parameters characterising the gas motion in the entire domain (for example, speed or concentration) have different scales and this usually hinders the smooth application of CFD modelling to the whole event in one shot. Splitting the phenomenon and, thus, creating a "Two-Steps" approach, has proven fruitful in finding the solution of the problem in due time.

However, as any theoretical model, once this approach is completely developed, it will need a validation through an experimental campaign. The objective of this work is to describe a suitable methodology to perform correctly scaled experiments of accidental high-pressure gas releases in oil and gas environments. The experimental setup that we designed is a reproduction of the real-size structure, on which the hazardous release may happen, that preserves the key feature of the environment: the congestion of spaces. A scaled platform, with a total of three decks, has been designed in conformity with existing examples installed in the Adriatic Sea, including volumes, cylinders and processing units as they can be found on-board. The mock-up platform has a scale of 1:10 with respect to the original and it is placed in a wind tunnel designed to create wind speed and flow conditions in similarity correlation to those that can occur in the Adriatic Sea. Finally, a release system has been designed to simulate the accidental leakage of gas in one of the decks of the scaled platform. The deck is conveniently equipped with sensors to capture velocity and concentration of the gas cloud that is formed after the release.

The final form of the experimental setup is based on scaling exercises, from the simplest geometrical scaling of the platform to the tricky scaling of freestream wind speed and release pressures and rupture diameters. The paper addresses all the scaling problems that have been dealt with and solved in the design of the experimental setup, to assure a physical consistency with the real-size phenomenon.

2. Background

Sometimes full-scale experiments are unfeasible because of cost, too large real-scale dimensions or safety limits because of the characteristics and the amount of dangerous substances involved in the experiment. A possible solution can be a reduced scale experiment, based on some scaling rules and on specific dimensionless parameters representative of the involved phenomena.

A scaling procedure strongly depends on the identification of the main physical phenomena that characterise the flow-field to be experimentally reproduced in a different scale. This identification is introduced, detailed and discussed in the case study reported hereafter.

2.1 Case study definition

We consider the accidental release of a highly pressurised flammable gas in an oil and gas offshore installation. The high-pressure gas release in the open environment leads to a highly under-expanded jet near the release point that includes regions with supersonic velocity; in fact, as the gas exits at a high pressure, it tends to rapidly adjusts to the ambient condition by means of some expansion–compression waves, and this interaction generates the typical highly under-expanded jet structure depicted in Figure 1.

Because of the strong expansion in the discharge ambient, the jet accelerates up to a supersonic velocity (Mach number, Ma $\gg$ 1), but after a certain distance, a normal shock occurs and the velocity immediately drops down to a subsonic value; this normal shock region is called Mach disk (Franquet *et al.*, 2015). This region is characterised by high discontinuities in all the flow field variables. The supersonic core region is called Mach cell, and it can extend for a length comparable to that of the release hole size, which is frequently around 1–5 cm in the oil and gas typical accidents (Vivalda *et al.*, 2018). Because of the supersonic velocity, the flow is compressible as Ma $\gg$ 0.3 (Munson *et al.*, 2010) and inertia dominated, therefore, buoyancy forces are negligible. The time scales involved are of the order of approximately 10 μs, that is, the time necessary for the Mach disk complete development (Tang *et al.*, 2017).

The phenomenon described above, involves only a small portion of the domain near the release point; in fact, as the distance from the leakage point increases, the flow slows down and a subsonic dispersion occurs in the main portion of the domain. As the initial high

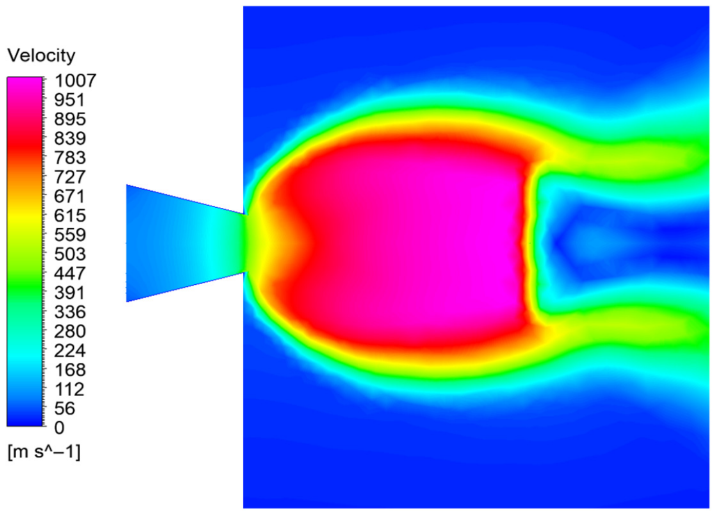

Figure 1.
Highly under-expanded jet structure

inertia is exhausted, the gas diffuses at low velocity (depending on the atmospheric wind velocities, in general approximately 5–6 m/s); therefore, the fluid can be considered incompressible (Ma $<$ 0.3) (Munson *et al.*, 2010) and gravity forces become relevant. The diffusion involves a large environment (an oil and gas platform can extend for tens of meters, see Figure 5), so that the order of magnitude of the time involved is of seconds, as the gas is slowly moving (approximately 5–6 m/s). In summary, two different flow regimes are involved, implying multi-physics and multi-scales issues. In addition, the wind presence must be carefully treated as it strongly affects the gas dispersion pattern: the wind velocity must be scaled considering its real distribution, i.e. logarithmic or exponential profile.

2.2 Scaling procedure

The facility that was designed (Tortora *et al.*, 2019) to experimentally investigate dispersion and diffusion of accidental gas releases in oil and gas platforms is an open-circuit wind tunnel with open-jet test section. As it is located in a large room within an already-existing building, some constraints guided the choice of the wind tunnel dimensions and characteristics. To ensure smooth, uniform and parallel flow in the test chamber, classical wind tunnel design criteria have been adopted in defining the geometry of the inlet contraction, of the test section and of the diffuser (Barlow *et al.*, 1999; Cattafesta *et al.*, 2010; Fang *et al.*, 2001; González Hernández *et al.*, 2013; Mehta and Bradshaw, 1979; Rodríguez Lastra *et al.*, 2013; Zanoun, 2018), but site-specific adaptations have been required by the inflow and outflow sections of the device.

Inflow conditions in the test chamber must be correctly scaled to replicate the real non-dimensional parameters. This is a typical problem that arises in wind tunnel experiments of pollutant dispersion in urban environment, but also in the scaled-down experimental analysis of hazardous gas releases in closed spaces such as tunnels or warehouses. The literature about this topic is quite extended, encompassing both papers specifically devoted to scaling procedures (Hall and Walker, 1997; Obasaju and Robins, 1998), guidelines for modelling atmospheric diffusion (Snyder, 1981; Snyder, 1985) or plume dispersion (Mavroidis *et al.*, 2003), experimental validation of CFD simulations of near-field pollutant dispersion (Gousseau *et al.*, 2011; Gupta *et al.*, 2012; Tominaga and Stathopoulos, 2013; Tominaga and Stathopoulos, 2016; Yassin, 2013) and experimental and/or numerical investigation of hydrogen releases in confined (Ekoto *et al.*, 2012; Houf *et al.*, 2012; Houf *et al.*, 2013), but also possibly ventilated (Giannissi *et al.*, 2015) spaces. A wealth of investigations has been carried out to simulate the atmospheric boundary layer in wind tunnels. The aim of the investigation can be broad and some examples are the effects of the wind on buildings and bluff bodies (Sheng *et al.*, 2018; Irwin, 2008) as well as the diffusion of pollutant (EPA, 1981).

Once the wind tunnel is designed, the dimensions of the analysed model (the plant or, as in this work, the deck of an offshore platform) must be scaled to be compatible with the test chamber dimensions, thus avoiding any interference related to the naturally growing boundary layer on the walls of the wind tunnel. At this point, the gas release scaling must be addressed.

The general approach for gaseous releases scaling is presented in Xing *et al.* (2014), where a reduced scale field experiment of CO_2 release was performed to validate numerical simulations. A positive side effect of a reduced scale experiment is the reduction of hazardous substance's need, reducing costs and significantly improving the experiments' safety.

The starting point for this scaling procedure is the *length scale* definition, which is based on the volume of the gas discharged into the environment [equations (1) and (2)]:

$$\text{Reduced} - \text{scale length}\, L = v^{\frac{1}{3}} \tag{1}$$

$$\text{Full} - \text{scale length}\ L = V^{\frac{1}{3}} \tag{2}$$

Once the ratio between the two length scales, i.e. the scaling factor S_c, is fixed, the geometry scaling follows [equation (3)]:

$$S_c = \frac{l}{L} \tag{3}$$

Then, the scaled volume can be derived from S_c:

$$\frac{v}{V} = S_c{}^3 \tag{4}$$

For the scaling of the reference velocities, respectively related to the wind and the gas release, three dimensionless parameters are considered: the density ratio, the Froude number and the Richardson number. The last two quantities are defined for both the wind (U) and the gas release (W) velocities:

$$\text{Density ratio}\ \frac{\rho_g}{\rho_a} \tag{5}$$

$$\text{Froude number (Fr)} \quad \frac{U^2}{gL} \quad \frac{W^2}{gL} \tag{6}$$

$$\text{Richardson number (Ri)} \qquad g\frac{\rho_g - \rho_a}{\rho_a}\frac{L}{U^2} \quad g\frac{\rho_g - \rho_a}{\rho_a}\frac{L}{W^2} \tag{7}$$

The *density ratio* accounts for buoyancy effects of the released gas in the air; the *Froude number* compares inertial and gravitational forces; and the *Richardson number* compares buoyancy and inertial forces. To satisfy the fluid dynamic similitude, equations (5) and (6) must keep the same value in full and reduced scale; once this is obtained, this is true also for equation (7).

Considering this procedure, the velocities scale according to:

$$\frac{w}{W} = \frac{u}{U} = S_c{}^{0.5} \tag{8}$$

The approach proposed by Xing *et al.* (2014) is very general and widely applicable. Nonetheless, this scaling procedure does not fit the present case study for several reasons:

- Xing *et al.* (2014) propose an approach suitable for dispersions characterised by release velocities (W) comparable to the wind velocity (U), while the initial pressure of accidental gas release here analysed guarantees a supersonic discharged flow, hence implying $W \gg U$.
- Xing *et al.*'s (2014) approach is developed for open-field experiments, while a wind tunnel is necessary to properly recreate the wind influence on a pressurised gas leakage in the offshore environment.

- The wind velocity scaling is badly addressed as it does not account for the typical profile of the atmospheric boundary layer, i.e. logarithmic or exponential wind profile; in fact, the wind velocity (*U*) is scaled using the same rules of the release velocity (*W*) scaling.

Another interesting scaling methodology is presented in Donat and Schatzmann (1999), where small-scale wind tunnel experiments for gaseous jets are studied; here, an approach based on a dimensional analysis is proposed, with the aim of obtaining a gas distribution in the small-scale experiment which is in fluid-dynamic similitude with the real scale one. As a first step, the authors defined their interest parameter, which is related to the gas concentration in the air; after that, they listed all the parameters affecting the interest variable. Applying dimensional considerations, they got the interest variable in a dimensionless form, and a set of dimensionless parameters to be kept constant to assure the similarity.

As a result, they obtained that to satisfy the similarity laws, all the dimensionless parameters involved must be kept constant. As in Xing *et al.* (2014), both the *density ratio* and the *Froude number* are considered for the scaling of the gas jet, but in addition also the turbulence of the gas jet is accounted for through the Reynolds number. A key difference with respect to Xing *et al.* (2014) is that the wind velocity scaling is addressed considering the atmospheric boundary layer, and a parameter which relates the gas jet velocity and the wind velocity is introduced. Even though this approach is more complete and involves more parameters, it is not applicable to the present case study in full for the following reasons:

- The number of dimensionless parameters involved is quite large; therefore, it is difficult to satisfy their equality between full and reduced scale model as many conflicts may arises. It is likely that some constraints will have to be relaxed.
- It does not fit the accidental release under consideration as it refers only to the dispersion phase; this corresponds to only a share of the entire phenomenon/domain here considered. This limit is well represented by the importance given to the *Froude number*, typically used in subsonic/dispersion models, where the gravity affects the phenomena evolution (e.g. the pollutant cloud dimension and position).

Kanda *et al.* (2006) introduced a dimensionless parameter that compares the inertia of the gas and wind flows analysing the car exhaust gas dispersion in the ambient air:

$$\frac{\text{Wind inertia}}{\text{Gas jet inertia}} \qquad \frac{\rho_a U^2}{\rho_g W^2} \tag{9}$$

The authors have assured the similarity in the experiment, keeping this parameter constant. In this case, there are no more difficulties arising from the managing of many dimensionless parameters: once the wind velocity is scaled using the method presented in Donat and Schatzmann (1999), the gas velocity scaling follows from equation (9). Nonetheless, the proposed parameter is not representative for the specific case of the relationship between the wind and the released pressurised gas: in fact, two orders of magnitude separate the wind and the gas velocities; moreover, given the square relationship, the proposed parameter will be small and largely affected by uncertainties.

Another parameter is given in Hall and Walker (1997); the authors proposed some scaling rules for a reduced scale field release of hydrogen fluoride. They stated that to comply with the similarity between real (full scale) and model (reduced scale) releases under wind condition, the following relation must be satisfied:

$$\frac{M_f}{U_H^2 L^2} = \frac{m_f}{u_H^2 l^2} \tag{10}$$

M_f and m_f are, respectively, the *source momentum flux* for real and model scale, i.e.:

$$M_f = W^2 A \frac{\rho_{g,R}}{\rho_a} \tag{11}$$

$$m_f = w^2 a \frac{\rho_{g,m}}{\rho_a} \tag{12}$$

with:

- W, w full and reduced release velocity;
- A, a full and reduced release hole area; and
- $\frac{\rho_{g,R}}{\rho_a}$, $\frac{\rho_{g,m}}{\rho_a}$ full and reduced scale density ratio (if the same gas is used, it is the same for both scales).

This seems to be the most complete dimensionless parameter for the scaling of a gas release, under wind conditions in a scaled geometry, as all the main parameters are involved: jet release velocity, density ratios, jet orifice dimensions, length scales and wind velocities.

The main advantages of this methodology are:

- The methodology is general, without any specific reference to subsonic or supersonic releases.
- The wind velocity profile is properly scaled (logarithmic profile) (see Section 3.2).
- A single equation links the geometry, the wind velocity and the supersonic release scaling procedures.

Hall and Walker's (1997) methodology was applied several times in the past to design experiments of gas leakages and dispersions at low speeds (Mavroidis *et al.*, 2003; Houf *et al.*, 2012; Ekoto *et al.*, 2012; Houf *et al.*, 2013); however, it cannot be directly applied to the proposed case study as the supersonic release presence makes it necessary to introduce some new hypotheses. An adaptation of the procedure to under-expanded jets is one of the purposes of the methodology presented in the next section of the paper. Moreover, for practical reasons linked to the design definition, it is useful to express the main scaling parameters as a function of the release pressure and the hole diameter.

3. Methods

The scaling of a real physical phenomenon is an important step for a more critical description and interpretation of the results emerging from the investigation. Moreover, not less important is the fact that the correct scaling allows a reliable reproduction of the real-scale physics.

The experimental reproduction on a reduced scale of the accidental release on a platform imposes specific requirements for the correct replica of the event. Namely, the scaling of the geometric parameters, of the atmospheric flow characteristics and the scaling related to the diffusion of the gas in the surrounding environment. The absolute similitude between two physical systems is sometimes impossible but reasonable assumptions in relation to the constraints of the problem allow reliable results from the scaled experiment. For example, for the purposes of the present study, the stratification of the atmospheric wind is not important and can be neglected. Moreover, the wind direction is assumed steady reproducing only the average real wind direction.

A valid guideline on the scaling rules to adopt in reduced scaled experiments is presented by Hall and Walker (1997). This work has been taken as reference for definition of the scaling parameters. In the following, a brief summary of the essential rules for the proper scaling of the boundary layer characteristics is carried out and then applied to the case study.

The complexity of the phenomena to be reproduced in the experiment arises mainly from the presence of two different flow regimes: the wind flow and the gas release flow; a scaling procedure method, which considers at the same time the two flows, is the objective of this work.

Before starting with the details of the scaling procedure, it is useful to summarise all the needed functional elements and relative systems and scaling procedures necessary to arrive to a correctly reduced scale experiment.

As we can see in Table 1, there are three main aspects to consider:

- the geometric scaling;
- the wind velocity scaling; and
- the release parameters scaling.

All these aspects are strictly correlated, and their dependencies can be appreciated in Figure 2.

The wind scaling depends on the geometric scaling, while the release scaling depends on both the other two.

3.1 Geometry scaling

As a wind tunnel will be used to reproduce the wind velocity, the dimensions of the platform mock-up strictly depend on the testing chamber size, which must contain the experimental mock-up and ensure that it is completely and correctly invested by the reproduced wind. For

Table 1. Functional systems and scaling procedures for the experimental campaign

Elements	Systems	Scaling procedure
An object to represent the real domain (offshore platform)	Platform mock-up	Geometry
An environment to reproduce the atmospheric wind condition	Wind tunnel	Wind velocity
A system to realise the gas release	Gas supply line	Release parameters

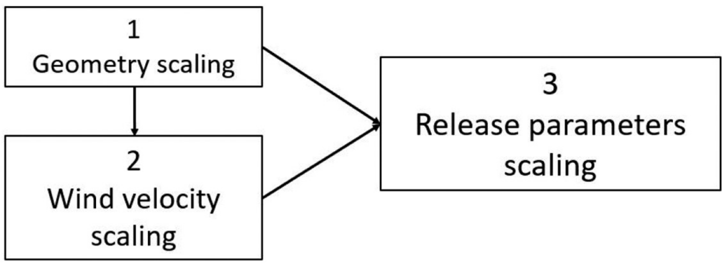

Figure 2. Scaling procedure scheme

this reason, a geometry scaling factor (S_c) of 1/10 is adopted to optimally use the available space of an existing structure, which will accommodate the wind tunnel. All the components present on the platform will be linearly scaled using the same factor.

3.2 Wind tunnel scaling

The design of the physical experiment in a wind tunnel to reproduce an accidental gas release in presence of atmospheric wind has to consider the properties of such flow field for the appropriate scaling.

The first modelling of the atmospheric boundary layer consists in the simulation of the mean velocity distribution in the lower part of the field. It must be remarked that equation (8) defines a velocity ratio between the reduced and the full scale considering only some requirements (density ratios and Froude and Richardson numbers) regardless the specific velocity distribution in the boundary layer. Using that equation, the resulting reduced-scale wind velocity u underestimates the velocity that would be present in the experiment if the velocity distribution in the full-scale atmospheric boundary layer were also considered.

The velocity distribution in the full-scale layer is greatly influenced by the roughness of the surface on which the wind flows. Taking into account also the aerodynamic roughness height, z_0 for the model and Z_0 for the full scale, the geometric scaling factor S_c applies for all length scales introduced. Therefore, it is possible to write:

$$S_c = \frac{l}{L} = \frac{h}{H} = \frac{z_0}{Z_0} \tag{13}$$

The lower part of the atmospheric boundary layer, the most interesting for the purpose of the present work, is characterised by a logarithmic profile velocity. Therefore, for the full scale and for the reduced scale, the velocity distributions can be, respectively, represented by:

$$\frac{U}{U_*} = \frac{1}{k} ln\left(\frac{Z}{Z_0}\right) \quad and \quad \frac{u}{u_*} = \frac{1}{k} ln\left(\frac{z}{z_0}\right) \tag{14}$$

where u_* and U_* indicate the friction velocities, while z and Z stand for the vertical coordinate from the surface (where upper-case letters stand for real-scale, and lower-case for reduced-scale).

To account for the higher velocity encountered in the real boundary layer with respect to the prescribed velocity from equation (8), the wind speed on the model at the distance h is increased considering the velocity on the model evaluated at higher distance H (Hall and Walker, 1997). The resulting velocity on the model than scales as:

$$\frac{u_H}{U_H} = S_c^{0.5}\left(1 - \frac{ln(S_c)}{ln\left(\frac{H}{Z_0}\right)}\right) \tag{15}$$

In case of studies inherent to gas dispersion, it is important to account for the turbulence characteristics of the atmospheric boundary layer. This means to reproduce on the scaled model at least the representative velocity fluctuation distribution across the boundary layer. For neutrally stable atmospheres, different relations are present in literature for the turbulent intensity distribution. A simple formula because of Panofsky and Dutton (1984) for the full scale is:

$$\frac{U'}{U_*} = \frac{kA}{ln(\frac{H}{Z_0})} \quad (16)$$

where U' is the *rms* value of the turbulent fluctuation while A is a constant specific for the fluctuation components (A = 2.5 for the longitudinal, A = 2 for the transversal and A = 1.3 for the vertical). It can be observed that for a fixed value of the length scale S_c the ratio $\frac{H}{Z_0}$ is the same for the reduced and full scale; therefore, the turbulent intensities distributions are also the same for the full scale and the reduced scale.

3.3 Release parameters scaling

The scaling procedure to be defined for the release phenomenon is the most challenging, because of the physical complexity of the gas jet here considered (highly under-expanded) and to its dependency on the geometric and wind velocity scaling. The first step is to identify the variables and the parameters characterizing the phenomenon. In general, for modelling a gas release it is important to define:

- position on the platform;
- direction;
- gas type;
- release hole diameter d_R (a circular hole is assumed); and
- release pressure p_{R0}.

The adaptation of the first two points to the reduced scale is trivial: the position in the platform is scaled according to S_c and the direction is the same of the full-scale case.

If the gas used in the experimental facility is the same as the one used for real industrial application, the reproduction of the accident will certainly be more reliable, and the scaling procedure will be simpler (see next paragraph). Unfortunately, in many cases, a different gas must be chosen for several reasons: cost, safety and traceability. In case the gas is different, several properties to be considered in the calculations are listed in Table 2, which summarises the symbols used for indicating full-scale and reduced-scale parameters, i.e. real case and model case.

A special attention must be given to the scaling of the release hole diameter (d_R) and pressure (p_{R0}). In the proposed analysis, only highly under-expanded gas jets are considered; the condition to be verified to have a highly under-expanded jet is given by (Franquet *et al.*, 2015) equation (17):

$$\frac{p_{R0}}{pa} > 7 \quad (17)$$

where p_a is the discharge ambient pressure.

Properties	R: Real case (full scale)	M: Model case (reduced scale)
Density [kg/m^3]	$\rho_{g,R}$	$\rho_{g,m}$
Adiabatic constant [–]	γ_R	γ_m
Critical ratio [–]	$R_{cr,R}$	$R_{cr,m}$
Gas constant [J/kg/K]	R_R	R_m

Table 2. Full and reduced scale gas properties

By means of the proposed scaling method, starting from a certain couple of parameters (p_{R0}, d_R) characterizing the full-scale case, it is possible to obtain the equivalent scaled couple of parameters (p_{m0}, d_m), assuring the fluid-dynamic similitude of the full and reduced scale (Figure 3).

The starting point is equation (10). Substituting equations (11) and (12) in equation (10), after some manipulations, it is possible to obtain:

$$\frac{w^2 \cdot a \cdot \rho_{g,m}}{W^2 \cdot A \cdot \rho_{g,R}} = \left(\frac{U_H}{u_H}\right)^2 \cdot (S_c)^2 \tag{18}$$

On the right-hand side of equation (18), the first term in the brackets is the inverse of the wind velocity scaling ratio, while the second term is the geometry scaling factor. On the left-hand side of equation (18), only the release parameters (release velocities, release hole areas and density of the released gases) appear. Therefore, it is important to notice that the ratio between the release parameters in the full and reduced scale depends on the wind and geometry scaling, which was the purpose of the work.

The next step is to re-write equation (18) as a function of the parameters of interest for this study case: the release pressures (p_{R0}, p_{m0}) and the hole diameters (d_R, d_m).

The densities $\rho_{g,m}$ and $\rho_{g,R}$ can be expressed in function of the release pressures. As only highly under-expanded jets are considered, i.e. equation (17) holds, the flow will be always chocked in correspondence of the exit section, depicted in Figure 4.

This means that the exit pressure can be evaluated by equation (19), and the exit velocity, i.e. the sonic velocity, by equation (20):

$$p_{R,exit} = R_{cr,R} \cdot p_{R0} \tag{19}$$

$$W = \sqrt{\frac{2\gamma_R \cdot R_R \cdot T_{R0}}{\gamma_R + 1}} \tag{20}$$

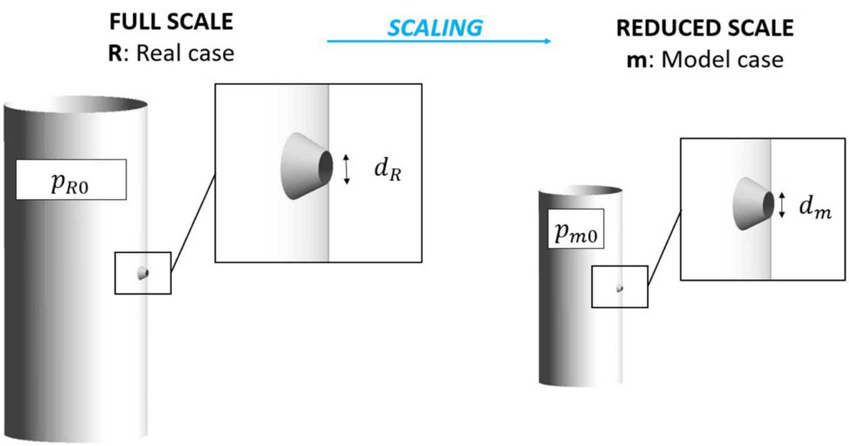

Figure 3.
Release parameters scaling

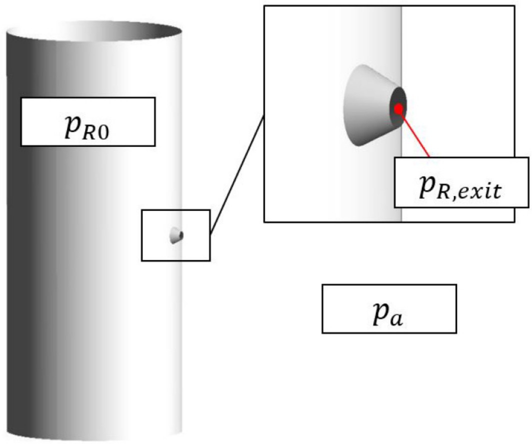

Figure 4.
Release exit section

where T_{R0} is the release temperature and $R_{cr,R}$ is the critical ratio, evaluated as is shown in equation (21):

$$R_{cr,R} = \left(\frac{2}{\gamma_R - 1}\right)^{\frac{\gamma_R}{\gamma_R - 1}} \tag{21}$$

Moreover, using the ideal gas law, the real-scale density at the exit of the hole can be evaluated with equation (22).

$$\rho_{g,R} = \frac{p_{R,exit}}{R_R \cdot T_{R,exit}} = \frac{R_{cr,R} \cdot p_{R0}}{R_R \cdot T_{R,exit}} \tag{22}$$

The same procedure can be followed to express the reduced scale density $\rho_{g,m}$ and the scaled release velocity w [equations (23) and (24)]:

$$\rho_{g,m} = \frac{p_{m,exit}}{R_m \cdot T_{m,exit}} = \frac{R_{cr,m} \cdot p_{m0}}{R_m \cdot T_{m,exit}} \tag{23}$$

$$W = \sqrt{\frac{2\gamma_m \cdot R_m \cdot T_{m0}}{\gamma_m + 1}} \tag{24}$$

Equations (23) and (24) are valid only if also in the reduced scale gas release the flow is chocked at the exit; this assumption must be verified at the end of the calculations.

With respect to temperature, the release temperature in the experimental facility is supposed to be the same as the one of the real case: $T_{R0} = T_{m0} = 300\,K$. Consequently, as $T_{R,exit} \sim T_{R0}$ and $T_{m,exit} \sim T_{m0}$, it is possible to deduce that $T_{R,exit} \sim T_{m,exit}$.

At the end, considering all the assumptions and substituting equations (20), (22), (23) and (24) in equation (18) we obtained:

$$\frac{a \cdot p_{m0}}{A \cdot p_{R0}} = \frac{2 \cdot \gamma_R \cdot (\gamma_m + 1) \cdot R_{cr,R}}{2 \cdot \gamma_m \cdot (\gamma_R + 1) \cdot R_{cr,m}} \cdot \left(\frac{U_H}{u_H}\right)^2 \cdot (S_c)^2 \tag{25}$$

Assuming a circular release hole, equation (25) can be written as:

$$\frac{d_m^2 \cdot p_{m0}}{d_R^2 \cdot p_{R0}} = \frac{2 \cdot \gamma_R \cdot (\gamma_m + 1) \cdot R_{cr,R}}{2 \cdot \gamma_m \cdot (\gamma_R + 1) \cdot R_{cr,m}} \cdot \left(\frac{U_H}{u_H}\right)^2 \cdot (S_c)^2 \tag{26}$$

Equation (26) can be used to find the couple of reduced scale values (p_{m0}, d_m) corresponding to a certain couple of real-scale parameters (p_{R0}, d_R). In the next section, some examples will be provided to better understand how equation (26) can be used.

4. Case study

The designed experimental facility is an open-circuit wind tunnel with rectangular open-jet test section, whose dimensions are 6.4 × 2.5 × 7.6 m in the spanwise, vertical and streamwise directions, respectively. Air enters the wind tunnel from the external environment through a 6.4 × 3.6 m opening and it flows through a series of screens and honeycomb layers to abate any residual fluctuations as well as large structures that may affect the flow quality within the test section.

Air acceleration is obtained through a 4.6 m long contraction with ratio equal to 1.44. The contraction ceiling height decreases according to two cubic polynomials (Zanoun, 2018). The equation of the convergent portion of the wind tunnel has been decided on the basis of preliminary simulations. In particular, the choice was deemed to be satisfactory in terms of the flow homogeneity and uniformity within the test section. After the test section, the air stream decelerates flowing through a 4 m long diffuser with 3.6° divergence angle. The flow, which is driven by ten axial fans disposed in two rows and located downstream the diffuser, is finally expelled in the environment through an opening in the ceiling.

The accidental scenario considered in this work is the release of pressurised methane on a natural gas extraction platform. For the purposes of this analysis, only the production deck is considered, and its dimensions are reported in Figure 5.

The considered dimensions are typical of a middle-class offshore production platform located in the Adriatic Sea. The full-scale platform (and consequently the reduced scale one) consists of three decks and four legs. The intermediate deck is the one considered for the experiments as a conservative choice, as this allows for the lowest dispersion of flammable gas because of the wind.

A release example characterised by the parameters presented in Table 3 is proposed here to apply the developed scaling methodology. The wind speed U in the real case at $H = 15$ m is assumed $U_H = 6$ m/s considering the statistical value in the region where the platform is mounted.

4.1 Application of the methodology

The physical extent of the uniform and parallel flow region in the test chamber imposes the geometry scaling factor choice. For the present case, the scale factor is set to $S_c = 1/10$, thus implying the mock-up dimensions equal to 3 × 2 × 0.5 m.

All the components on the deck will be reproduced with the correspondent scaled dimensions in the mock-up. Moreover, the experimental facility will need concentration and flow sensors for the measurements and monitoring; the sensors will be hidden inside the objects present in the geometry to limit to a minimum the interference with the flow-field inside the model. For safety reasons, pure methane cannot be used in the experiment to avoid the possibility of having flammable concentrations, so an air-methane mixture is preferred. The composition is:

$$2.2\%\ CH_4 + 97.8\%\ air$$

As we can see, the mixture contains a very poor methane concentration, hence all the properties of this gas can be well approximated with the air ones ($\gamma_m = 1.4$, $R_{cr,m} = 0.5283$).

Knowing the geometrical scaling S_c, the wind velocity can be obtained by means of equation (15), where we assumed H/Z_o = 150. The resulting scaled speed at the reference scaled height (1.5 m) in the wind tunnel u_H is equal to 2.77 m/s. Despite the scaling procedure permits to evaluate the correct scaled velocity value at the scaled height, a uniform velocity distribution will be fixed in the test section; this condition seems to be a good approximation of the real case, as the real platform height (15 m) is high enough to make the platform invested by the upper part of the logarithmic wind profile, which is almost constant.

At this point, the scaled release parameters can be evaluated. The preliminary step is to calculate d_m using S_c:

$$d_m = S_c \cdot d_R = 0.001\ m$$

At this point, all the data to be used in equation (26) are available and a p_{m0} = 6.35 bar is obtained.

It is fundamental to check if the critical flow assumption is verified also in the experiment, considering that the discharge ambient pressure p_a is 1 bar. Using the critical flow criterion, the critical conditions are verified:

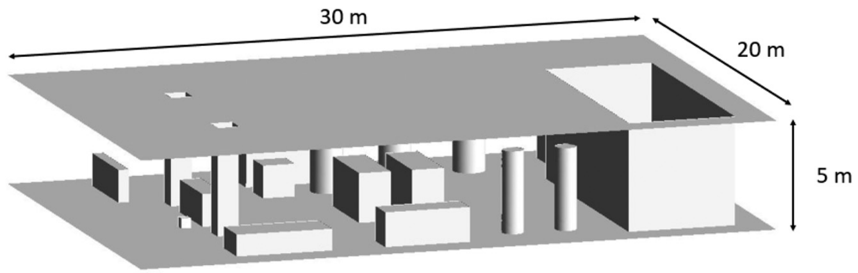

Figure 5. Platform deck dimensions

Table 3. Case study example parameters

Wind velocity U_H [m/s]	6
Release pressure p_{R0} [bar]	30
Release temperature T_{R0} [K]	300
Release hole diameter d_R [m]	0.01
Methane adiabatic constant γ_R	1.3
Methane critical ratio $R_{cr,R}$	0.5448

$$\frac{p_a}{p_{m0}} = \frac{1}{6.35} \ll R_{cr,m} \tag{27}$$

therefore, the procedure is correctly applied.

As a further example, one may keep all the parameters as constant with the exception of the release pressure which is increased to $p_{R0} = 10$ bar; applying the scaling procedure, $d_m = 0.001$ m and $p_{m0} = 2.13$ bar are obtained.

The last check is about the chocking condition in the model case, which is satisfied:

$$\frac{p_a}{p_{m0}} = 2.13 < R_{cr,m} \tag{28}$$

It is important to stress that this result stems from the application of the theoretically implied model. In practice, a real gas supply line will be used to perform the experimental gas release. It is then important to account for the uncertainties because of the operations and to the physical characteristics of the line (e.g. pressure drops). For example, assuming a conservative 15% error on the desired p_{m0}:

$$p_{m0} = 2.13 \pm 15\% \; bar$$

which, in the worst case, means $p_{m0} = 1.8$ bar; in this situation, the critical condition in the reduced scale release is not satisfied as shown by equation (29):

$$\frac{p_a}{p_{m0}} = \frac{1}{1.8} > R_{cr,m} \tag{29}$$

The solution to be able to perform a meaningful test, in this case, is to relax the condition of a precise geometric scaling of the release diameter using a smaller value, like $d_m = 0.0008$ m. If this value is used in equation (26), a value of $p_{m0} = 3.33$ bar is obtained. This value, regardless of the characteristics and the uncertainties of the gas supply line, ensures the chocking condition of the flow and the validity of the scaling procedure.

5. Conclusions

In this work, we describe a scaling procedure used to perform reduced-scale experiments to reproduce high-pressure accidental gas releases in open industrial environments.

Scaling laws for subsonic gas releases or gas dispersion have been well addressed in numerous past works and related literature, but some additional issues arise when the gas leakage produces a supersonic jet. Here we adapt the existing scaling procedures to a specific case study: the accidental high-pressure gas release followed by an under-expanded jet.

The complete scaling procedure involves three main aspects:

- geometric scaling;
- wind velocity scaling; and
- gas release parameters scaling.

The geometric scaling is addressed simply defining a scaling factor, S_c, while the other two require more sophisticated procedures. The wind velocity is scaled considering the real wind velocity profile (logarithmic profile) and the geometric scaling factor S_c. A wind tunnel has been designed and it is under construction at the time of writing. The experimental results will be used to validate the numerical simulation tools used to predict the behaviour of the accidental gas leakage. Finally, the scaling of the gas release parameters is the most difficult point. Our approach consists in expressing all the needed relations in terms of release pressure and rupture diameter and in deriving an equation that provides the couple of scaled rupture diameter and release pressure that maintain fluid-dynamic similitude with the real values.

The main outcome of the work is the possibility to perform a scaling that contains all the important elements characterising the phenomenon that must be scaled and experimentally reproduced. Once the geometry is scaled according to a certain scaling factor S_c, the scaled wind velocity follows and at the end, the scaled release parameters can be derived coherently with the previous steps, i.e. a fully integrated scaling procedure is developed.

A limit of the methodology is represented by the fact that it is developed for under-expanded jets, i.e. only this kind of releases can be considered. Anyway, in most industrial plants, the typical components pressure ranges from 5 bar to very high values (for example, 150 bar or more), so that the method has a wide applicability range.

References

Barlow, J.B., Rae, W.H. and Pope, A. (1999), *Low-Speed Wind Tunnel Testing*, 3rd ed., John Wiley and Sons.

Carpignano, A., Corti, T., Uggenti, A.C. and Gerboni, R. (2017), “Modelling of a supersonic accidental release in oil and gas offshore: characterisation of a source box”, *GEAM: Geoingegneria Ambientale e Mineraria*, Vol. 152 No. 3, pp. 58-64.

Cattafesta, L., Bahr, C.J. and Mathew, J. (2010), “Fundamentals of wind-tunnel design”, in Blockley, R. and Shyy, W. (Eds), *Encyclopedia of Aerospace Engineering*, John Wiley and Sons.

Donat, J. and Schatzmann, M. (1999), “Wind tunnel experiments of single-phase heavy gas jets released under various angles into turbulent cross flows”, *Journal of Wind Engineering and Industrial Aerodynamics*, Vol. 83 Nos 1/3, pp. 361-370.

Ekoto, I.W., Houf, W.G., Evans, G.H., Merilo, E.G. and Groethe, M.A. (2012), “Experimental investigation of hydrogen release and ignition from fuel cell powered forklifts in enclosed spaces”, *International Journal of Hydrogen Energy*, Vol. 37 No. 22, pp. 17446-17456.

Environmental Protection Agency (EPA) (1981), “Summary of the 1981 EPA (Environmental Protection Agency) national performance audit program on source measurements”, available at: www.epa.gov (accessed 29 September 2020).

Fang, F.M., Chen, J.C. and Hong, Y.T. (2001), “Experimental and analytical evaluation of flow in a square-to-square wind tunnel contraction”, *Journal of Wind Engineering and Industrial Aerodynamics*, Vol. 89 Nos No. 3-4, pp. 247-262.

Franquet, E. Perrier, V. Gibout, S. and Bruel, P. (2015), “Free underexpanded jets in a quiescent medium: a review”, available at: www.researchgate.net/publication/281629558_Review_on_the_underexpanded_jets (accessed 18 December 2019).

Giannissi, S.G., Shentsov, V.D., Melideo, Cariteau, B., Baraldi, D., Venetsanos, A.G. and Molkov, V. (2015), “CFD benchmark on hydrogen release and dispersion in confined, naturally ventilated space with one vent”, *International Journal of Hydrogen Energy*, Vol. 40 No. 5, pp. 2415-2429.

González Hernández, M.A., Moreno López, A.I., Jarzabek, A.A., Perales, J.M., Wu, Y., and., and Xiaoxiao, S. (2013), “Design methodology for a quick and low-cost wind tunnel”, in Ahmed, N.A. (Ed.), *Wind Tunnel Designs and Their Diverse Engineering Applications*, IntechOpen.

Gousseau, P., Blocken, B., Stathopoulos, T. and van Heijst, G.J.F. (2011), "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*, Vol. 45 No. 2, pp. 428-438.

Gupta, A., Stathopoulos, T. and Saathoff, P. (2012), "Wind tunnel investigation of the downwash effect of a rooftop structure on plume dispersion", *Atmospheric Environment*, Vol. 46, pp. 496-507.

Hall, D.J. and Walker, S. (1997), "Scaling rules for reduced-scale field releases of hydrogen fluoride", *Journal of Hazardous Materials*, Vol. 54 Nos 1/2, pp. 89-111.

Houf, W.G., Evans, G.H., Ekoto, I.W., Merilo, E.G. and Groethe, M.A. (2013), "Hydrogen fuel-cell forklift vehicle releases in enclosed spaces", *International Journal of Hydrogen Energy*, Vol. 38 No. 19, pp. 8179-8189.

Houf, W.G., Evans, G.H., Merilo, E., Groethe, M. and James, S.C. (2012), "Releases from hydrogen fuel-cell vehicles in tunnels", *International Journal of Hydrogen Energy*, Vol. 37 No. 1, pp. 715-719.

Irwin, P.A. (2008), "Bluff body aerodynamics in wind engineering", *Journal of Wind Engineering and Industrial Aerodynamics*, Vol. 96 Nos 6/7, pp. 701-712.

Kanda, I., Uehara, K., Yamao, Y., Yoshikawa, Y. and Morikawa, T. (2006), "A wind-tunnel study on exhaust gas dispersion from road vehicles-Part I: velocity and concentration fields behind single vehicles", *Journal of Wind Engineering and Industrial Aerodynamics*, Vol. 94 No. 9, pp. 639-658.

Mavroidis, I., Griffiths, R.F. and Hall, D.J. (2003), "Field and wind tunnel investigations of plume dispersion around single surface obstacles", *Atmospheric Environment*, Vol. 37 No. 21, pp. 2903-2918.

Mehta, R. and Bradshaw, P. (1979), "Design rules for small low speed wind tunnels", *The Aeronautical Journal*, Vol. 83 No. 827, pp. 443-453.

Munson, B., Young, D., Okiishi, T., Huebsch, W. and Rothmayer, A. (2010), "Compressible flow", *Fluid Mechanics*, 7th ed., Chapter 11, Wiley, pp. 599-660.

Obasaju, E.D. and Robins, A.G. (1998), "Simulation of pollution dispersion using small scale physical models – an assessment of scaling options", *Environmental Monitoring and Assessment*, Vol. 52 Nos 1/2, pp. 239-254.

Panofsky, H.A. and Dutton, J.A. (1984), *Atmospheric Turbulence: models and Methods for Engineering Applications*, Wiley.

Rodríguez Lastra, M., Fernández Oro, J.M., Galdo Vega, M., Blanco Marigorta, E. and Santolaria, M. (2013), "Novel design and experimental validation of a contraction nozzle for aerodynamic measurements in a subsonic wind tunnel", *Journal of Wind Engineering and Industrial Aerodynamics*, Vol. 118, pp. 35-43.

Sheng, J.X., Jacob, D.J., Turner, A.J., Maasakkers, J.D., Sulprizio, M.P., Bloom, A.A., Andrews, A.E. and Wunch, D. (2018), "High-resolution inversion of methane emissions in the Southeast US using SEAC4RS aircraft observations of atmospheric methane: anthropogenic and wetland sources", *Atmospheric Chemistry and Physics*, Vol. 18.

Snyder, W.H. (1981), "Guidelines for fluid modelling of atmospheric diffusion", EPA-600/8-81-009, United States Environmental Protection Agency.

Snyder, W.H. (1985), "Fluid modeling of pollutant transport and diffusion in stably stratified flows over complex terrain", *Annual Review of Fluid Mechanics*, Vol. 17 No. 1, pp. 239-266.

Tang, X., Asahara, M., Koichi Hayashi, A. and Tsuboi, N. (2017), "Numerical investigation of a high pressure hydrogen jet of 82 MPa with adaptive mesh refinement: the starting transient evolution and Mach disk stabilization", *International Journal of Hydrogen Energy*, Vol. 42 No. 10, pp. 7120-7134.

Tominaga, Y. and Stathopoulos, T. (2013), "CFD simulation of near-field pollutant dispersion in the urban environment: a review of current modeling techniques", *Atmospheric Environment*, Vol. 79, pp. 716-730.

Tominaga, Y. and Stathopoulos, T. (2016), “Ten questions concerning modeling of near-field pollutant dispersion in the built environment”, *Building and Environment*, Vol. 105, pp. 390-402.

Tortora, A., Uggenti, A.C., Gerboni, R., Moscatello, A., Bono, N., D’Ambrosio, D. and Iuso, G. (2019), “Modelling and monitoring gas dispersion in an oil & gas platform: project definition and design of a validation experiment”, *Offshore Mediterranean Conference and Exhibition 2019 in Ravenna*.

Vivalda, C., Gerboni, R. and Carpignano, A. (2018), “A practical approach to risk-based gas monitoring system design for oil and gas offshore platforms”, *Proceedings of the 14th Probabilistic Safety Assessment and Management Conference*.

Xing, J., Liu, Z., Huang, P., Feng, C., Zhou, Y., Sun, R. and Wang, S. (2014), “CFD validation of scaling rules for reduced-scale field releases of carbon dioxide”, *Applied Energy*, Vol. 115, pp. 525-530.

Yassin, M.F. (2013), “A wind tunnel study on the effect of thermal stability on flow and dispersion of rooftop stack emissions in the near wake of a building”, *Atmospheric Environment*, Vol. 65, pp. 89-100.

Zanoun, E.S. (2018), “Flow characteristics in low-speed wind tunnel contractions: simulation and testing”, *Alexandria Engineering Journal*, Vol. 57 No. 4, pp. 2265-2277.

Further reading

Armitt, J. and Counihan, J. (1968), “The simulation of the atmospheric boundary layer in a wind tunnel”, *Atmospheric Environment (1967)*, Vol. 2 No. 1, pp. 49-71.

Hancock, P.E. and Hayden, P. (2016), “Wind-tunnel simulation of stably stratified atmospheric boundary layers”, *Journal of Physics: Conference Series*, Vol. 753, p. 032012.

Hancock, P.E. and Hayden, P. (2018), “Wind-tunnel simulation of weakly and moderately stable atmospheric boundary layers”, *Boundary-Layer Meteorology*, Vol. 168 No. 1, pp. 29-57.

Saathoff, P., Gupta, A., Stathopoulos, T. and Lazure, T. (2009), “Contamination of fresh air intakes due to downwash from a rooftop structure”, *Journal of the Air and Waste Management Association (1995)*, Vol. 59 No. 3, pp. 343-353.

Snyder, W.H. (1972), “Similarity criteria for the application of fluid models to the study of air pollution meteorology”, *Boundary-Layer Meteorology*, Vol. 3 No. 1, pp. 113-134.

Sluman, T.J., van Maanen, H.R.E. and Ooms, G. (1980), “Atmospheric boundary layer simulation in a wind tunnel using air”, *Applied Scientific Research*, Vol. 36 No. 4, pp. 289-307.

Corresponding author
Alberto Moscatello can be contacted at: alberto.moscatello@polito.it