WIND INFLUENCE IN NUMERICAL ANALYSIS OF NSHEVS PERFORMANCE

Wojciech Węgrzyński and Grzegorz Krajewski

[1] Building Research Institute (ITB)
w.wegrzynski@itb.pl
[2] Building Research Institute (ITB)
g.krajewski@itb.pl

Abstract. This paper treats on the subject of including wind as a boundary condition in CFD analysis used in Fire Safety Engineering. Adverse wind effect is observed mostly on the performance of natural smoke and heat ventilators – and often is included in numerical studies performed by engineers. This paper provides with general guidelines on the Computational Wind Engineering, as well as to relevant references. Paper emphasizes on the necessity of building large enough domain and performing an angle sensitivity analysis, to determine the worst wind conditions for the vents. Only then the fire related CFD/CWE coupled analysis may be performed, with reasonable and believable results.

# 1.	SHORT INTRODUCTION TO COMPUTATIONAL WIND ENGINEERING

## 1.1.	Introduction

Computational Wind Engineering (CWE) is primarily defined as the use of Computational Fluid Dynamics (CFD) for wind engineering applications [3]. In last 50 years, this application went a transition, from emerging field into an increasingly established field in research, practice, and design. CWE is used in prediction of wind comfort, pollution, dispersion or loading on a building [13]. The scale of this analysis may be considered as metrological microscale, but due to the complexity of flows around buildings, especially in urban areas, its requirements may be regarded as one or two orders of magnitude higher, than the requirements for typical CFD application in fire related science. In Fire Safety Engineering (FSE) CFD is used to predict the movement of smoke and heat within building structures. The meeting point between both can be found in complex applications of natural smoke and heat ventilators (NSHEV). NSHEV remove the smoke and heat from the building into the atmosphere due to the small difference in the density of hot gasses inside of the building and atmosphere. For such devices, the wind is an important design factor, that may determine the performance of the system, and as such define the conditions inside in case of the fire.

The interface between CWE and FSE is not well described in the literature; there are insufficient data on validation of such coupled analysis. The most researched areas of CWE are (i) simulations of the Atmospheric Boundary Layer (ABL); (ii) bluff-body aerodynamics; (iii) turbulence modelling and numerical techniques; (iv) verification and validation in CFD for urban physics and wind engineering [21][4].

## 1.2.	Areas of Interest

The CWE focus lies within the metrological microscale and the lowest part of the ABL. In this field of the atmosphere, the Coriolis force is lowest and does not influence the flow within the model in a way; that would justify modelling it. Scales of the atmospheric phenomena that are investigated range from fractions of centimetre's (turbulence), metre's (building wakes and thermal flows) to kilometres (convection, urban heat islands) [4]. The time scale of the phenomena does also scale from fractions of seconds (dissipation of turbulence) to hours and days (metrological phenomena). The medium size phenomena are usually directly simulated, while larger become boundaries of the model and smaller parametrized solutions. For more detailed description of the scales of the analysis, please refer to [27].

The main areas of interest of CWE are (i) structural wind engineering; (ii) pedestrian-level wind and urban flows; (iii) natural ventilation of buildings; (iv) wind-driven rain and snow transport [3].

In FSE, following main areas of the use of CFD may be distinguished: (i) assessment of tenable conditions in the building in fires; (ii) growth and spread of the fire (also forensic); (iii) thermal effects of fires on structures and heat transfer (iv) ventilation system performance.

2. MEETING POINT BETWEEN CWE AND FSE

2.1. NSHEVs design practice

Modern design methodologies [6,23,33] condition the required amount of smoke ventilators on the size of the design fire and supply air solution. These methods origin in the work of Thomas [30] and others [15,18], who applied Bernoulli's law to the flow of hot smoke and combustion products from the burning compartment to surrounding. Methods presented below in Eq. 1 [6] and 2-3 [33] require vast knowledge of the designer on the fire itself. Variables that are boundary conditions for the analysis are the depth of smoke layer, the temperature of the smoke or mass flows within the compartment. Even with this detailed information, the result of the calculation is just a general overview of what is the approximate total area of all ventilators required to protect the compartment, but without any information on individual features of these ventilators (e.g. aerodynamic free area, opening angle).

$$A_{vtot}C_v = \frac{M_l T_l}{[2\rho_{amb}^2 g d_l \theta_l T_{amb} - \frac{M_l^2 T_l T_{amb}}{(A_i C_i)^2}]^{\frac{1}{2}}} \quad (1)$$

where: A_{vtot} – total required area of smoke dampers [m²], C_V – discharge coefficient of smoke dampers, M_l – mass flow of smoke [kg/s], T_l – average temperature of smoke [K], ρ – ambient air density [kg/m³], g – gravity [N/kg], Θ – increment of smoke temperature [K], T_{amb} – ambient temperature [K], A_i – total area of inlets [m²], C_i – discharge coefficient of inlets [-].

$$A_{vtot} = \frac{\dot{V}}{\bar{c}_{v0}} \sqrt{\frac{T_{amb}}{2g\Theta d - \frac{1}{\bar{c}_{v0,in}^2} w_i^2 T_l}} \quad (2)$$

$$A_i = \frac{1}{w_i}(\frac{\dot{m}_{Pl}}{\rho_{amb}} - \dot{V}) \quad (3)$$

where: A_{vtot} – total required area of smoke dampers [m²], $C_{v0,ab}$ – discharge coefficient of smoke dampers, $C_{v0,in}$ – discharge coefficient of air inlets, T_{amb} – ambient temperature [K] , T_P – average temperature of smoke [K], g – gravity [N/kg], w_i – flow velocity referred to the geometrical surface area of inlets [m/s], Θ – increment of smoke temperature [K], A_i – total area of inlets [m²], m_{pl} – mass flow of smoke in fire plume [kg/s], ρ_{zu} – ambient air density [kg/m³], V – volume flow of air supplied by mechanical means [m³/h].

Despite the complexity of the calculation procedure, it still does not account the wind influence on the system performance, besides the introduction of a discharge coefficient for the ventilator.

2.2. Estimation of C_v

Natural smoke and heat exhaust ventilators (NSHEV) are considered safety equipment of the building, and as such are under appropriate supervision, enshrined in the mandate 109 of European Commission [37]. Under Regulation 305/2011 [32] their production, certification, and distribution in member countries of European Union, is governed by the provisions of the harmonized standard EN 12101-2 [7]. The NSHEV performance is dependent on the wind; its negative influence is traditionally stated in the form of discharge coefficient (C_v), varying in value between 0,20 to 0,80. Note must be taken, that this coefficient is different, than ones estimated in pioneering work by Prahl and Emmons [25], as its value is determined always for the same conditions (as described below), and can be considered independent from the Reynolds number of the flow. This is one of the reasons, why practical implementation of this value is difficult in hand calculation methods.

The area of NSHEV multiplied by the discharge coefficient is referred to, as the aerodynamic free area, and is considered as the effective area of an NSHEV through which the flow of hot smoke occurs in wind conditions. As it is the only parameter describing the "performance" of the device, the manufacturers of natural smoke ventilators often improve the value of the discharge coefficient by mounting additional elements, such as fairings, directing jets or increasing the opening angle of the device. Besides the increase in the C_v value, the global efficiency of such solutions in a building remains unknown. By harmonized standard EN 12101-2 [7], the discharge coefficient of a ventilator is evaluated with (C_{vw}) and without (C_{v0}) the side wind, Figure 1.
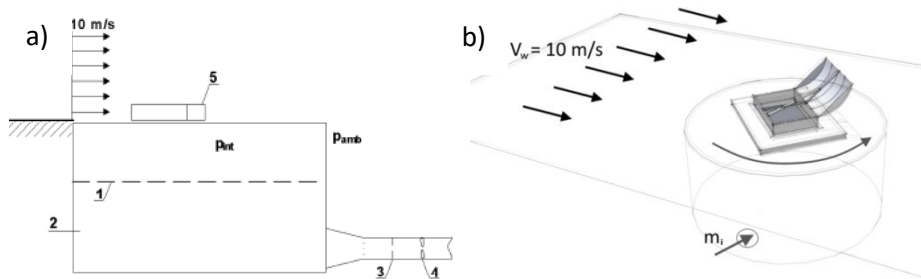
Key: **1** – screen, **2** – settling chamber, **3** –volume flow measurement, **4** – fan, **5** – ventilator

Figure 1. Scheme of the test chamber (1a) and a 3D visualisation (1b) of the setup used in the discharge coefficient assessment [3]

In EN 12101-2 test [3], the ventilator is mounted on the top of the settling chamber, which is located beneath the wind tunnel floor in a way, that its roof is in the line of the wind tunnel floor. The air velocity in the tunnel should be 10 m/s (±0,5 m/s), and the turbulence intensity should not exceed 20% (10% at a certain height). The standard does not limit the uncertainty of the measurement, but it must be sufficient to measure the mentioned limiting values. The wind attack angles are altered by the rotation of the settling chamber, together with the ventilator mounted on it. According to EN 12101-2

the value of discharge coefficient for a single pressure point, at the most difficult wind attack angle, is determined by following the formula (4). Next, with the use of mathematical regression the value of received coefficient can be determined for similar devices of the same producer, depending on their opening angle, the height of the ventilator and the deflector and the aspect ratio of the ventilator throat area.

$$C_V = \frac{\dot{m}_{ing}}{A_{v,test}\sqrt{2\rho_{air}\Delta p_{int}}} \qquad (4)$$

where: m_{ing} - mass flow into the settling chamber, A_v – total area of the tested ventilator, ρ_{air} – ambient air density, Δp_{int} – pressure difference between settling chamber and the wind tunnel.
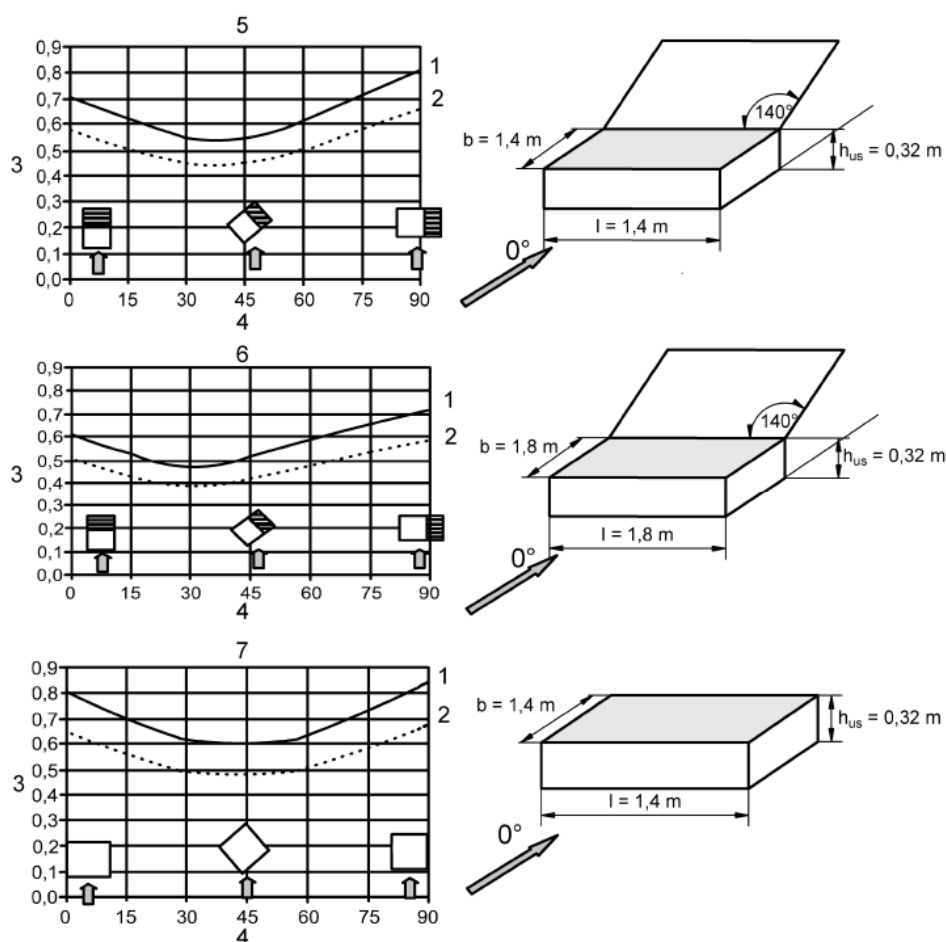
1 – upper limit, 2 – lower limit, 3 - Cv value of the tested ventilator, 4 – wind attack angle (V = 10 m/s) 5 – test case with 0,32 m high ventilator with 1,40 m x 1,40 m dimensions and 140° opening angle, 6 - test case with 0,32 m high ventilator with 1,80 m x 1,80 m dimensions and 140° opening angle, 7 - test case with 0,32 m high ventilator with 1,40 m x 1,40 m with no closure

Figure 2. Reference test cases for wind tunnels validation, Annex B of prEN 12101-2 [8]

2.3.   Issues with practical use of C_v

Determination of a single C_v value for a natural ventilator may be misleading when the performance of a whole system is assessed. This performance is also dependent on the wind velocity, angle and location of inlets to the building. A comprehensive study on this was presented in the past [17], and a result of an extensive numerical study done by the authors is currently in press [34]. The performance of multiple combinations of natural ventilators and the wind was presented as a part of Case Study 2 at SFPE Conference in Warsaw, 2016 [35].
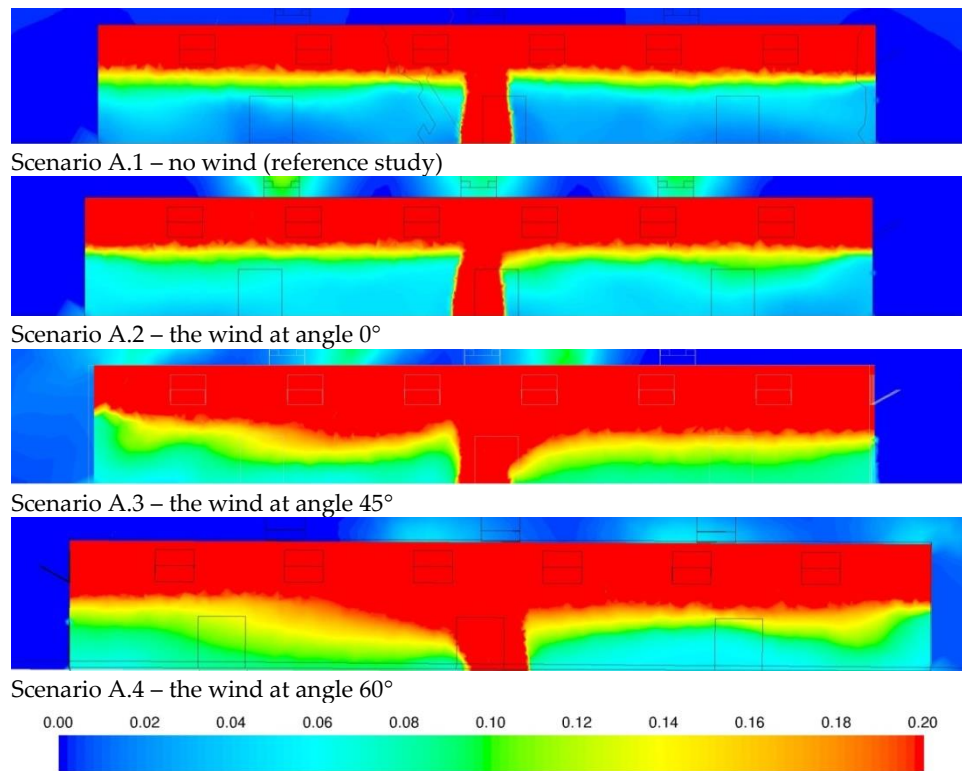
Scenario A.1 – no wind (reference study)

Scenario A.2 – the wind at angle 0°

Scenario A.3 – the wind at angle 45°

Scenario A.4 – the wind at angle 60°

| 0.00 | 0.02 | 0.04 | 0.06 | 0.08 | 0.10 | 0.12 | 0.14 | 0.16 | 0.18 | 0.20 |

Figure 3. Comparison of local mass concentration of smoke (0,00 – 0,20 g/m³ and more) in section through the building for various wind angles [34]

Angle	Wind velocity u_{ref} [m/s/]	roof mounted smoke ventilators	roof mounted ventilators with deflectors	wall mounted ventilators on back façade	wall mounted ventilators on front façade
-	0 (reference)	33,25	34,6	23,8	-
0	4	30,4	31,8	22,9	8,75
45°	4	27,6	29,1	23,5	11,8
60°	4	25,4	27,1	22,2	13,7
90°	4	29,7	29,7	19,0	18,5
60°	8	18,3	20,8	23,7	

Table 1. Mass flow of smoke [kg/s] in the analysis [34]

3. COMPLEX APPROACH TO ESTIMATION OF NSHEVS PERFORMANCE

3.1. Common issues with modelling wind for FSE

In [35] authors determined three different approaches to numerical modelling, that are common in FSE/CWE coupling:
a) model simplified to include only the interior of the building, outlets modelled as pressure boundary conditions;
b) model simplified to include the interior of the building, and nearest exterior, outlets modelled as an opening in the walls along with their most important features, pressure boundaries at the edges of the domain; insufficient size of the domain for CWE;
c) model suitable for wind engineering, with exterior domain large enough to not influence the flow around the building, outlets modelled in details as physical openings with all of their features, pressure boundaries at the edges of the domain with velocity boundary including logarithmic wind profile on velocity inlet.

The first approach (a) is sufficient only for most basic, preliminary analysis, without the wind. The simplification in the modelling of the inlets and outlets will strongly influence the performance of NSHEVs. The second method (b) is valid for NSHEV performance analysis, but without any wind interaction. This method can be used for checking the environmental conditions connected to the evacuation process inside the building, but the designer must add a margin of safety to the results, as in wind conditions they may be significantly worse. The introduction of wind velocity in such approach will lead to increased wind at the walls, due to flow compression, and may not be considered as a valid wind analysis. The third approach (c) is used for precise evaluation of NSHEVs performance in wind conditions, following the requirements of Chapter 4, for different wind angles.
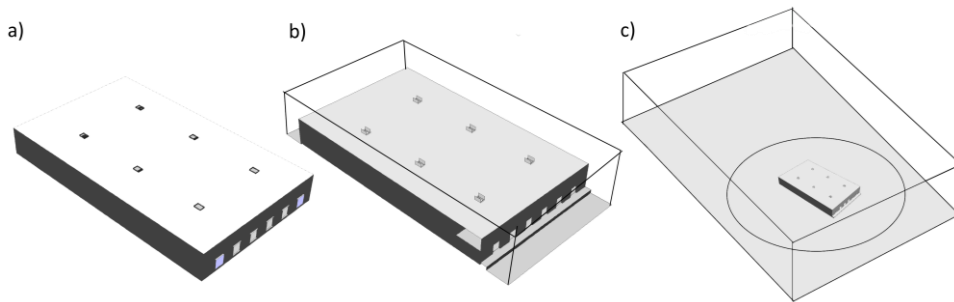

Figure 4. Three approaches to modelling NSHEVS in FSE/CWE coupling [35]

3.2. Workflow

FSE/CWE coupling is a tedious task. It is impossible to guess, which wind direction will be the worst condition for the fire, as local structures or geographical features of the terrain may strongly influence this. For simple buildings and simple combinations of buildings, hand calculation methodology presented in

Eurocode 1-4 may be sufficient [9], but for more complex structures and urban environment, CFD analysis is necessary. According to pedestrian level wind guidelines and urban flow guidelines (Chapter 4.1), not less than 12 wind angles should be verified.

Investigation of multiple wind attack angles in a transient, fire simulation is extremely time and resource consuming task. As a simplification, the analysis may be decoupled into two steps.

The first step, wind analysis, is to evaluate the wind influence over external features of the building, without a fire (and often without interior model). This analysis relies on statistical wind data for the area, in which the building is located. Numerical model used in the study should include the building analysed in a domain, according to Chapter 4.2, and allow its free rotation. The analysis itself is steady-state and averaged conditions are estimated. Results of such study are investigated with respect to wind pressure coefficient in areas on which elements of the natural smoke exhaust system are located.
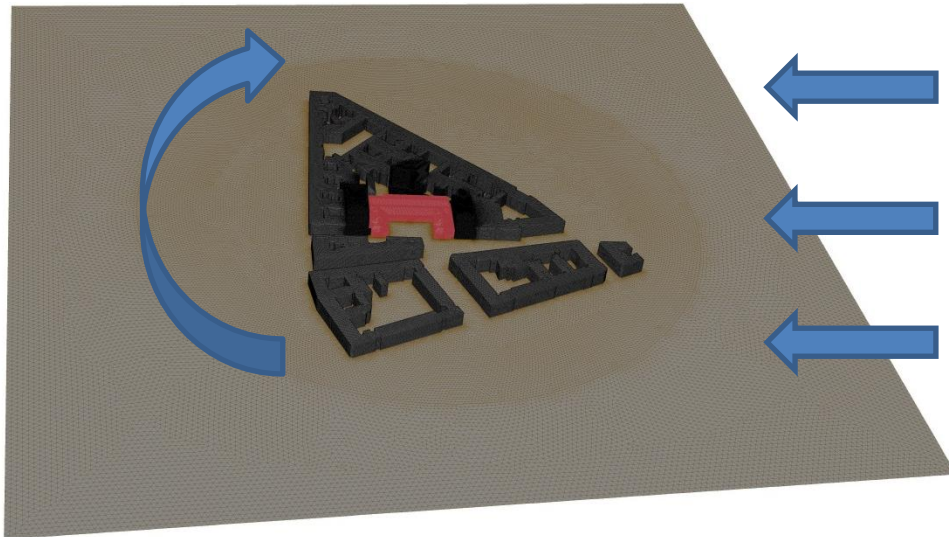

Figure 5. Fragment of the numerical model used in a wind influence analysis (own work)
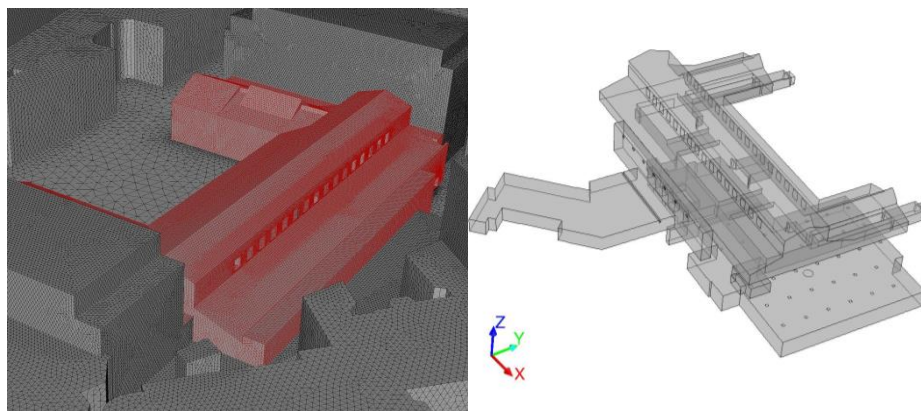

Figure 6. Close up of the building in the middle of Figure 5, exterior with visible unstructured numerical mesh (left) and interior of the model (right) (own work)

Worst case scenario is usually one with the highest pressure on the ventilators, highest under-pressure in areas where inlet air openings are located or the case in which highest air velocity inside the building is observed. It is possible that this analysis will yield multiple scenarios to be evaluated in further CFD studies. Although fairly straightforward, this analysis may be quite difficult, mainly because of the need for simultaneous simulation of effects of both large-scale buildings and other environmental obstacles and small scale elements of natural smoke and heat exhaust system and other items that may affect the aerodynamics of the roof. Once the worst case scenario (or scenarios) are determined, step two of the analysis is performed, which is a transient simulation with a fire within the building in the worst wind conditions.
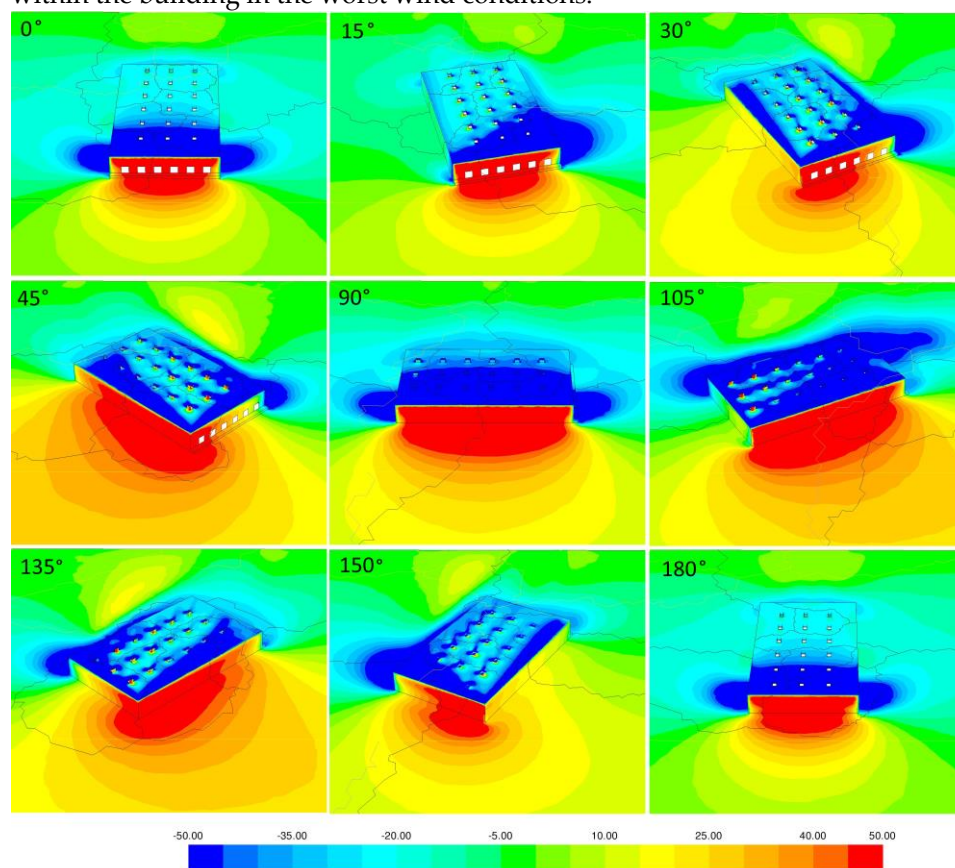

Figure 7. Nondimensional pressure coefficient at a building subject to the wind, u_{ref} = 10 m/s at various wind angles [35]

4.   SUMMARY OF RELEVANT GOOD PRACTICE

4.1.   CWE good practice guidelines

Multiple good practice guidelines are available for CWE. Please relate to these documents when performing wind-related studies for the evaluation of NSHEV performance: [3,4,11,13,12,31].

It must be noted, that most of CWE guidelines consider first-order numerical schemes as not appropriate for CFD modelling, and at least second-order schemes are required [16,31].

When referring to FSE/CWE coupled analysis, the engineer should take into account the recommendations of (i) pedestrian-level wind and (ii) natural ventilation analysis. The first ones describe the workflow required to obtain a valid solution of airflow at "human" height above the ground – which are valid for areas in which both inlets on the ground-level and outlets on the roofs are placed. The second type of guidelines will relate to the coupling of internal and external flows in the buildings, although it must be noted that this field is still under rapid development, and solutions are not available for many problems. Another thing to consider, are the recommendations related to urban area flows, which are mainly included in the description of computational domain size and the mesh requirements in the far-field.

4.2. Size of the domain and level of details

Many rules of CWE originate from decades of testing in wind tunnels. One of such rules is that the blockage ratio in the cross-section at which flow occurs should not be larger than 3% [11]. The blockage ratio can be described as the proportion of cross-section of the building to the cross-section of the domain, in a plot perpendicular to the flow. Illustration of the blockage ratio is presented in Figure 8.
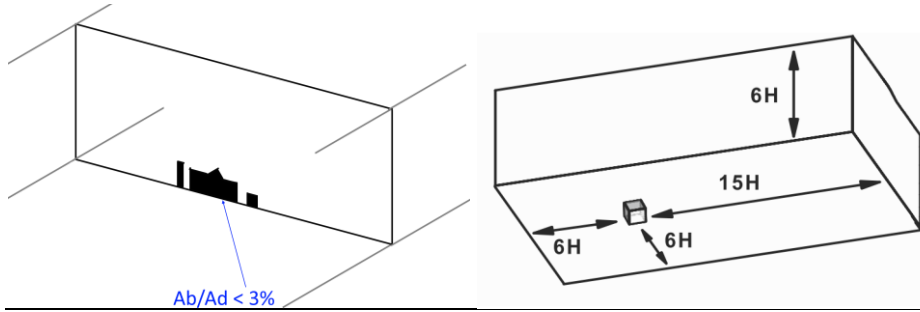

Figure 8. Blockage ratio (left) and correct dimensions of numerical domain (right) [13]

The inlet, lateral and top boundary should be at least $5H_{max}$ away from the group of explicitly modeled buildings, where H_{max} is the height of the tallest building. The reason for this is to limit the error caused by the modeling technique on the airflow velocity in building proximity – too small domain will cause strong artificial acceleration. Because the dynamic pressure of the airflow is determining the performance of NSHEVs, this error is significant for the natural ventilation performance assessment [5]. The outflow boundary should be at least 10 [31] to 15 [13] H_{max} away from the group of explicitly modeled buildings, to allow for full wake flow development. The size of the domain is presented in Figure 8, and a sample of a large-scale numerical model is shown in Figure 10.

The main difficulty of application of CWE into FSE comes in the requirement for high-quality representation of roof and its details in the analysis – resulting in

massive scale differences between smallest and largest elements in the model. By the experience of authors, who performed multiple optimization studies on natural ventilators, the details larger than 5 cm should be represented in the ventilator model, as these elements will have an influence on the discharge coefficient of the ventilator greater than 0,02. A sample of detailed models of ventilators is shown in Figure 9.
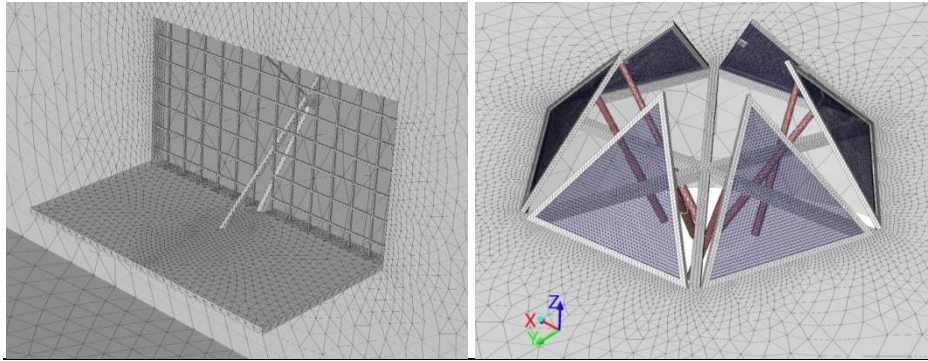

Figure 9. Sample numerical models of complex natural ventilators used in the estimation of C_{vv} value (own work)
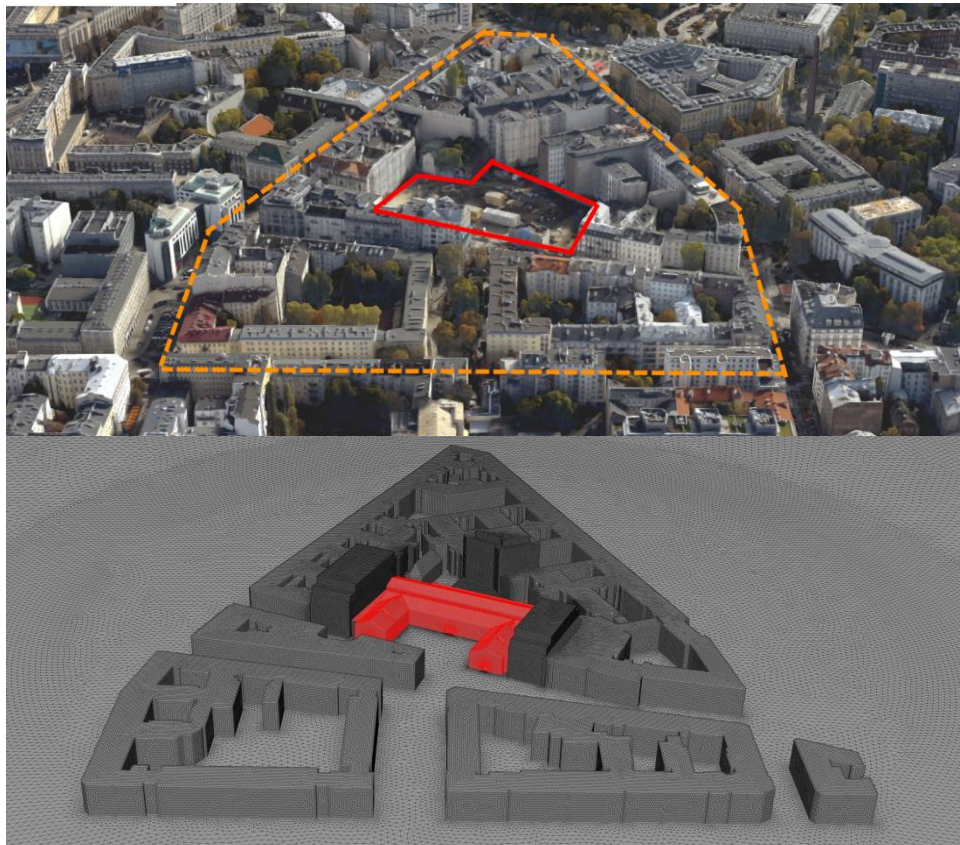

Figure 10. Aerial photograph of Warsaw (upper picture, source: Google Earth) and the numerical domain in the model (bottom picture, source: own work)
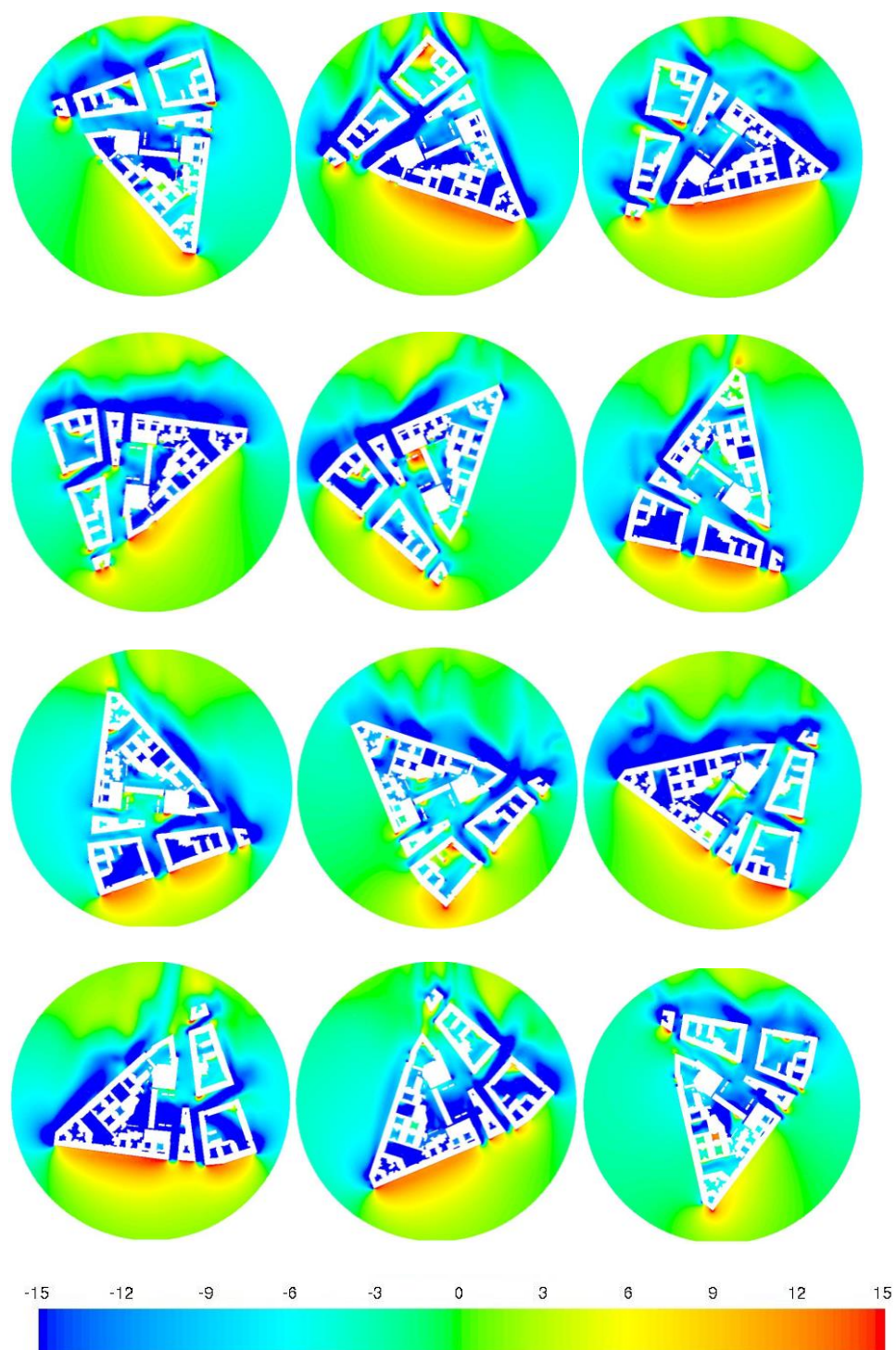
Figure 11. Nondimensional pressure coefficient at buildings subject to the wind,
u_{ref} = 4 m/s at various wind angles [35]

4.3. Time discretization

The length of discrete time step should follow the good practice rules for FSE CFD. If FDS solver is chosen, the time step length is evaluated by the solver, basing on CFL criterion, and largely dependent on the size of numerical mesh [20]. In the case of other solvers, it should provide proof of convergence, although the default convergence criteria may be too loose.

Two types of solvers used for numerical modelling:

- Steady-state solver;
- Unsteady (transient) solver.

By assumption, the steady state solution represents the solution of the flows in the volume independent of the time. In FSE most of the analysis is carried as transient, as the fire itself is a transient phenomenon. Another thing is that thermal effects of the flow are typically neglected in the wind-related analysis, however, in the fire-related analysis, thermal effects will have a strong influence on the solution. The heat transfer between fire, smoke and building partitions, even for a constant fire, may take hours to stabilize. This is the reason, why obtaining a converged solution in steady state even for the fire with a constant size is difficult (or impossible). CFD user must note that with LES turbulence modelling, the solution is transient by assumption.

Performing all wind-related studies as transient would require immense computational power, the workaround this is to use the decoupled approach presented in Chapter 3. First, multiple steady state analysis may be performed to find the worst case scenarios, and then these are verified with transient analysis and in fire conditions.

4.4. Space discretization

Two characteristics of high-quality computational grids are (i) sufficient overall grid resolution and (ii) quality of the computational cells regarding shape (including skewness), orientation and stretching ratio.

For FSE, our typical best practice for mesh generation is more than sufficient for CWE; typical mesh size for FSE is determined by the root of Froude number, which describes the ratio of the flow inertia to an external force (in this case the buoyant force) of the flow and is often considered as the criterion of similarity between various fires. In most cases, the mesh size will fit between 10 to 20 cm. For the CWE approach, at least ten cells per cube root of the building volume should be used, and at least ten cells between buildings. In the ground layer, at least five elements shall be placed at the height at which velocity is critical – a rule that may be extended to both pedestrian level inlets to the building, and roof level outlets. These are usually met by high-quality fire oriented mesh. Also, second essential mesh requirement – quality of the computational cells regarding shape is explicitly met in solvers such as FDS, where Cartesian, structured mesh is used. In other solvers, this should be a part of the individual analysis. Unstructured meshes (such as a tetrahedral mesh) will increase truncation errors and cause issues with convergence.

The only factors that are difficult to meet in typical FSE related CFD analysis with Cartesian mesh are the orientation of the mesh and the growth rate factor.

First is necessary for the angle sensitivity study; that should include 12 various angles. It is hard to rotate the model, and not introduce errors, even with the saw-tooth limiting sub-models. Recommended growth ratio of elements should not exceed $1,30 : 1$ [13], while for Cartesian mesh the recommended growth ratio is $2 : 1$ [19]. Due to this, a mesh sensitivity study should be performed to assess the influence of mesh transitions on the flow. For unstructured meshes, a growth function with a ratio less than $1,30 : 1$ is necessary to create sufficient domain. A relevant guide on high-level mesh generation for a coupled outdoor and indoor analysis is available in [16].

4.5.   Turbulence modelling

For a detailed information on the turbulence models used in CWE, choice of the model and their validation, please refer to [2,3,12,22].
For applications in wind engineering, the common closure models are:
- Steady Reynolds-averaged Navier-Stokes (RANS);
- Unsteady Reynolds-averaged Navier-Stokes (URANS);
- Large Eddy Simulation (LES);
- Hybrid URANS/LES approach (e.g. DES).

In fire related study, two approaches are dominant:
- Reynolds-averaged Navier-Stokes (RANS);
- Large Eddy Simulation (LES).

RANS model family consists of multiple models, and the best fit for CWE are Realizable k-ε and RNG k-ε models [11]. The standard k-ε model should not be used for this application.

LES is intrinsically superior regarding physical modelling to both RANS and URANS. It is suitable for simulating three specific characteristics of the turbulent bluff body in urban physics: three-dimensionality of the flow, unsteadiness of the large-scale flow structures and anisotropy of turbulent scalar fluxes. However, 3D steady RANS remains the main CFD approach and has a satisfactory degree of success. An interesting concept is to combine LES and RANS into one model with massively separated flows, in which large vortices are resolved by LES, while small by RANS approach. This is often referred as Detached Eddy Simulation model (DES) [29], but despite the simplification it still requires similar computational power as LES models [2].

It must be noted, that the popularity of LES modelling in FSE is connected with the popularity of fire-oriented CFD code Fire Dynamics Simulator [19], while many other software, such as ANSYS Fluent [1], STAR-CCM+ [28], Phoenics [10], Smartfire [14], use or allow user to choose RANS approach, or other turbulence models. The choice of the turbulence model is often connected with the selection of the software and the hardware limitations of the user. LES modelling requires finer mesh, and its solution has higher computational cost – in some studies, it is estimated, that this cost is at least one order larger than for steady RANS [4]. This is why in CWE RANS remains the main CFD approach, and has a satisfactory degree of success. A summary of validation studies on both RANS and LES modelling can be found in [3].

4.6. Description of wind boundary condition

The description of the wind boundary condition (inlet to the model) is an implementation of ABL model into CFD. This implementation usually requires the knowledge of two parameters – the upstream aerodynamic roughness length and vertical profile for mean velocity and turbulence properties [4]. The first one can be done using Davenport classification, as updated by Wieringa (and often referred as Davenport-Wieringa model) [36]. Aerodynamic roughness length will determine the velocity profile and the turbulence parameters of the flow within the domain and essentially will drive the movement of air in the proximity of the building. Blocken describes five spatial areas, out of which four should lie within the computational domain, for which the roughness should be specified, along with the requirements of [4].

For the second parameter, the wind velocity profile, the most used profiles in RANS CFD in urban physics and wind engineering are those by Richards and Hoxey [26]. It must be noted, that field measurements and reduced-scale wind-tunnel measurements of turbulence intensity do not always yield a profile of turbulent kinetic energy that is constant with height in the surface layer (assumption of the Richards and Hoxey model). An alternative way is provided by Tominaga [31], to obtain k(z) from a wind-tunnel experiment. Otherwise, a specific profile for the streamwise turbulence intensity should be provided.

Provision of a reliable ABL model for LES modelling may be difficult, as the turbulent behaviour of the wind cannot be simplified in the k and ε parameters, but has to be explicitly modelled in a transient approach. Some guidelines on this are available in [24].

5. CONCLUSIONS

The introduction of high-quality Computational Wind Engineering into a Fire Safety Engineering workflow is a challenging and arduous task. The computational requirements and the cost of coupled analysis may be considered a one or two orders of magnitude higher than the requirements for the same analysis, but only of the building interior. However, there are cases in which this extraordinary expense is of worth – the design of breakthrough structures, which use natural ventilation of unrivalled efficiency and close to none environmental cost. Such systems are the essence of "green" and "sustainable" development. With our current knowledge, we are not bound to traditional, simplified, methodologies for the evaluation of such systems performance, which opens a niche for highly optimised solutions.

Multiple problems occur when CWE good practice guidelines meet typical FSE workflow: the mesh generation, order of numerical schemes, the size of the domain, angle sensitivity, introduction of the wind as a boundary condition to name a few. Some of them may be resolved by a skilled engineer who builds the case and the model; other may require systematic approach (especially LES modelling in Cartesian meshes). The community has to take an active part in resolving these issues, if we hope to have a reliable tool for CWE/FSE coupled

model in short future, at the quality of the FSE oriented CFD models, as we do have today.

6. BIBLIOGRAPHY

[1]   ANSYS. (2014) ANSYS Fluent 14.5.0 - Technical Documentation.
[2]   Argyropoulos, C.D. and Markatos, N.C. (2015) Recent advances on the numerical modelling of turbulent flows. *Applied Mathematical Modelling*, Elsevier Inc. **39**, 693–732. https://doi.org/10.1016/j.apm.2014.07.001
[3]   Blocken, B. (2014) 50 years of Computational Wind Engineering: Past, present and future. *Journal of Wind Engineering and Industrial Aerodynamics*, **129**, 69–102. https://doi.org/10.1016/j.jweia.2014.03.008
[4]   Blocken, B. (2015) Computational Fluid Dynamics for urban physics: Importance, scales, possibilities, limitations and ten tips and tricks towards accurate and reliable simulations. *Building and Environment*, Elsevier Ltd. **91**, 219–45. https://doi.org/10.1016/j.buildenv.2015.02.015
[5]   Blocken, B., Stathopoulos, T. and van Beeck, J.P.A.J. (2016) Pedestrian-level wind conditions around buildings: Review of wind-tunnel and CFD techniques and their accuracy for wind comfort assessment. *Building and Environment*, Elsevier Ltd. **100**, 50–81. https://doi.org/10.1016/j.buildenv.2016.02.004
[6]   BSI. (2003) BS 7974 Part 4: Components for smoke and heat control systems. Functional recommendations and calculation methods for smoke and heat exhaust ventilation systems, employing steady-state design fires. Code of practice.
[7]   CEN. (2003) EN 12101-2:2003 Smoke and heat control systems. Specification for natural smoke and heat exhaust ventilators.
[8]   CEN. (2015) prEN 12101-2 Smoke and heat control systems Part 2: Specification for natural smoke and heat exhaust ventilators.
[9]   CEN. (2005) EN 1991-1-4:2005 Eurocode 1: Actions on structures - Part 1-4: General actions - Wind actions.
[10] CHAM. (2005) PHOENICS Overview - CHAM Technical Report: TR 001 [Internet].
[11] Franke, J., Hirsch, C., Jensen, A.G., Krüs, H.W. and Schatzmann, M., Westbury, P.S., et al. (2004) Recommendations on the use of CFD in wind engineering. *Proceedings of the International Conference on Urban Wind Engineering and Building Aerodynamics COST Action C14, Impact of Wind and Storm on City Life Built Environment,*.
[12] Franke, J. Introduction to the Prediction of Wind Loads on Buildings by Computational Wind Engineering (CWE). *Wind Effects on Buildings and Design of Wind-Sensitive Structures*, Springer Vienna, Vienna. p. 67–103. https://doi.org/10.1007/978-3-211-73076-8_3
[13] Franke, J., Hellsten, A., Schlünzen, H. and Carissimo, B. (2007) Best practice guideline for the CFD simulation of flows in the urban environment. COST Office Brussels.
[14] FSEG. Smartfire Introduction [Internet].
[15] Hansell, G.O. (1993) Heat and mass transfer process affecting smoke control in atrium buildings. South Bank University.
[16] van Hooff, T. and Blocken, B. (2010) Coupled urban wind flow and indoor natural ventilation modelling on a high-resolution grid: A case study for the Amsterdam ArenA stadium. *Environmental Modelling and Software*, Elsevier Ltd. **25**, 51–65. https://doi.org/10.1016/j.envsoft.2009.07.008
[17] Marchant, E.W. (1984) Effect of wind on smoke movement and smoke control systems. *Fire Safety Journal*, **7**, 55–63. https://doi.org/10.1016/0379-7112(84)90008-0
[18] McCaffrey, B.J. (1979) Purely Buoyant Diffusion Flames: Some Experimental Results. NBSIR 79-1910.
[19] McGrattan, K., Hostikka, S., McDermott, R., Floyd, J., Weinschenk, C. and Overholt, K. (2016) Fire Dynamics Simulator User's Guide, Sixth Edition. https://doi.org/10.6028/NIST.SP.1019
[20] McGrattan, K., McDermott, R., Floyd, J., Hostikka, S., Forney, G. and Baum, H. (2012) Computational fluid dynamics modelling of fire. *International Journal of Computational Fluid Dynamics*, **26**, 349–61. https://doi.org/10.1080/10618562.2012.659663
[21] McGrattan, K. and Miles, S. (2016) Modeling Fires Using Computational Fluid Dynamics (CFD). *SFPE Handbook of Fire Protection Engineering*, Springer New York, New York, NY. p. 1034–65. https://doi.org/10.1007/978-1-4939-2565-0_32
[22] Murakami, S. (1998) Overview of turbulence models applied in CWE–1997. *Journal of Wind Engineering and Industrial Aerodynamics*, **74–76**, 1–24. https://doi.org/10.1016/S0167-

6105(98)00004-X

[23]   NFPA. (2015) NFPA 204 Standard for Smoke and Heat Venting 2015 Edition.

[24]   Porté-Agel, F., Wu, Y.-T., Lu, H. and Conzemius, R.J. (2011) Large-eddy simulation of atmospheric boundary layer flow through wind turbines and wind farms. *Journal of Wind Engineering and Industrial Aerodynamics*, **99**, 154–68. https://doi.org/10.1016/j.jweia.2011.01.011

[25]   Prahl, J. and Emmons, H.W. (1975) Fire induced flow through an opening. *Combustion and Flame*, **25**, 369–85. https://doi.org/10.1016/0010-2180(75)90109-1

[26]   RICHARDS, P.J. and HOXEY, R.P. (1993) Appropriate boundary conditions for computational wind engineering models using the k-ε turbulence model. *Computational Wind Engineering 1*, Elsevier. p. 145–53. https://doi.org/10.1016/B978-0-444-81688-7.50018-8

[27]   Schlünzen, K.H., Grawe, D., Bohnenstengel, S.I., Schlüter, I. and Koppmann, R. (2011) Joint modelling of obstacle induced and mesoscale changes-Current limits and challenges. *Journal of Wind Engineering and Industrial Aerodynamics*, **99**, 217–25. https://doi.org/10.1016/j.jweia.2011.01.009

[28]   Siemens. Star-CCM+ [Internet].

[29]   Spalart, P.R. (2009) Detached-Eddy Simulation. *Annual Review of Fluid Mechanics*, **41**, 181–202. https://doi.org/10.1146/annurev.fluid.010908.165130

[30]   Thomas, P.H., Hinkley, P.L., Theobald, C.R. and Simms, D.L. (1963) Investigations into the flow of hot gases in roof venting. Fire Res. Tech. Pap. No 7. London.

[31]   Tominaga, Y., Mochida, A., Yoshie, R., Kataoka, H., Nozu, T., Yoshikawa, M. et al. (2008) AIJ guidelines for practical applications of CFD to pedestrian wind environment around buildings. *Journal of Wind Engineering and Industrial Aerodynamics*, **96**, 1749–61. https://doi.org/10.1016/j.jweia.2008.02.058

[32]   UE. (2011) Regulation (EU) No 305/2011 of The European Parliament and the Council of 9th March 2011 laying down harmonised conditions for the marketing of construction products and repealing Council Directive 89/106/EEC. 5–43.

[33]   VDI. (2006) VDI 6019 Blatt 1 Ingenieurverfahren zur Bemessung der Rauchableitung aus Gebäuden Brandverläufe, Überprüfung der Wirksamkeit.

[34]   Węgrzyński, W. and Krajewski, G. Influence of wind on natural smoke and heat exhaust system performance in fire conditions (in press). *Journal of Wind Engineering and Industrial Aerodynamics*,.

[35]   Węgrzyński, W., Krajewski, G. and Sulik, P. (2016) Case Study 2 – Production and Storage Building (Poland). *11th Conference on Performance-Based Codes and Fire Safety Design Methods*, SFPE, Warszawa. https://doi.org/10.13140/RG.2.1.3677.4640

[36]   Wieringa, J. (1992) Updating the Davenport roughness classification. *Journal of Wind Engineering and Industrial Aerodynamics*, **41**, 357–68. https://doi.org/10.1016/0167-6105(92)90434-C

[37]   M/109 Mandate to CEN and CENELEC concerning the execution of standardisation work for harmonised standards on fire alarm/detection, fixed fire-fighting, fire and smoke control and explosion suppression products.