Codes and standards on computational wind engineering for structural design: State of art and recent trends

Original
Codes and standards on computational wind engineering for structural design: State of art and recent trends / Bruno, L; Coste, N; Mannini, C; Mariotti, A; Patruno, L; Schito, P; Vairo, G. - In: WIND AND STRUCTURES. - ISSN 1226-6116. - 37:2(2023), pp. 133-151. [10.12989/was.2023.37.2.133]

Publisher:
TECHNO-PRESS

Published
DOI:10.12989/was.2023.37.2.133

Wind and Structures, *Vol. 37, No. 2 (2023) 133-151*
https://doi.org/10.12989/was.2023.37.2.133

Codes and standards on computational wind engineering for structural design: State of art and recent trends

Luca Bruno*[1,8], Nicolas Coste[2,8], Claudio Mannini[3,8], Alessandro Mariotti[4,8],
Luca Patruno[5,8], Paolo Schito[6,8] and Giuseppe Vairo[7,8]

[1]*Department of Architecture and Design, Politecnico di Torino, viale Mattioli 39, 10131 Torino, Italy*
[2]*Optiflow Company, 160 Chem. de la Madrague-Ville, 13015 Marseille, France*
[3]*Department of Civil and Environmental Engineering, University of Florence, Via di Santa Marta, 3 – 50139 Firenze, Italy*
[4]*Department of Civil and Industrial Engineering, University of Pisa, Largo Lucio Lazzarino, 2 – 56122 Pisa, Italy*
[5]*Department of Civil, Chemical, Environmental, and Materials Engineering, Viale del Risorgimento 2, 40126 Bologna, Italy*
[6]*Department of Mechanical Engineering, Politecnico di Milano, via La Masa 1 - 20156 Milano, Italy*
[7]*Department of Civil Engineering and Computer Science, University of Rome "Tor Vergata", via del Politecnico 1 – 00133 Roma, Italy*
[8]*Special Interest Group in Computational Wind Engineering (ANIV-CWE),*
Italian Association for Wind Engineering, www.aniv-iawe.org/aniv-cwe

(*Received October 9, 2022, Revised February 27, 2023, Accepted July 5, 2023*)

Abstract. This paper first provides a wide overview about the design codes and standards covering the use of Computational Wind Engineering / Computational Fluid Dynamics (CWE/CFD) for wind-sensitive structures and built environment. Second, the paper sets out the basic assumptions and underlying concepts of the new Annex T "Simulations by Computational Fluid Dynamics (CFD/CWE)" of the revised version "Guide for the assessment of wind actions and effects on structures" issued by the Advisory Committee on Technical Recommendations for Constructions of the Italian National Research Council in February 2019 and drafted by the members of the Special Interest Group on Computational Wind Engineering of the Italian Association for Wind Engineering (ANIV-CWE). The same group is currently advising UNI CT021/SC1 in supporting the drafting of the new Annex K – "Derivation of design parameters from wind tunnel tests and numerical simulations" of the revised Eurocode 1: Actions on structures – Part 1-4: General actions – Wind actions. Finally, the paper outlines the subjects most open to development at the technical and applicative level.

Keywords: best practices; codes and standards; Computational Wind Engineering; Eurocode PrEN 1991-1-4:2021–Annex K; Guide CNR-DT 207-R1/2018-Annex T

1. Introduction

Piero Pozzati, one of the founding fathers of Structural Design in Italy, stated in his last academic lesson "*An excess of technical standards involves various inconveniences* […]: *depletion of designer's autonomy and creativity; difficulty in discerning what really matters; feeling of being relieved of responsibility; difficulty in understanding the reasoning that underly the rules; considering rules as algorithms the thinking is no longer called on to explain and justify*" (Pozzati 1992). Such a statement looks still topical 30 years later in general (Paya-Zaforteza and Garlock 2021), and, in particular, twice more relevant to the codification of Computational Wind Engineering (CWE) for the evaluation of wind loads on structures.

Does CWE need to face its wide and increasing use within the structural design practice through codification? Who shall be the main target of such a codification activity? What shall be its purpose? Does codification increase the risk of a non-conscious and uncritical use of CWE by practitioners?

The present paper attempts to answer those questions, as tackled by the Special Interest Group on Computational Wind Engineering of the Italian Association for Wind Engineering (ANIV-CWE, www.aniv-iawe.org/aniv-cwe). ANIV-CWE includes some of the most experienced Italian native computational wind engineers, coming from both academic and non-academic sectors. In particular, the paper develops in the wake of the support given by ANIV-CWE to the drafting of the informative annexes on CWE of two recent recommendations and standards: Annex T "Simulations by Computational Fluid Dynamics" of the "Guide for the assessment of wind actions and effects on structures" issued by the Advisory Committee on Technical Recommendations for Constructions of the Italian National Research Council in 2018 (CNR-DT R1-207/2018), and the upcoming new edition of Annex K "Derivation of design parameters from wind tunnel tests and numerical simulations" of Eurocode 1 - Actions on Structures - Part 1-4: General Actions - Wind Actions (PrEN 1991-1-4:2021, Ricciardelli 2023), whose approved final version is expected to appear in 2024.

The present paper, conceived to be equally suitable for both specialized and non-specialized readers, starts from the

---

*Corresponding author, Ph.D. Professor
E-mail: luca.bruno@polito.it

http://www.techno-press.com/journals/was&subpage=5

ISSN: 1226-6116 (Print), 1598-6225 (Online)

analysis of the past and current CWE practice for structural design, and traces the evolution in time of its codification. Readers interested in a broader and more general overview of the CWE can refer to the number of excellent review papers published so far (e.g., Murakami 1990, 1997, Stathopoulos 2002, Blocken 2014, Tamura and Van Phuc 2015, Potsis *et al*. 2023). Then, an overview of the guiding principles which inspired the drafting of the aforementioned informative annexes is provided, together with a summary of their content. Additional considerations which emerged in the drafting activity are reported, aiming at highlighting points which appear to be of particular importance or most open to significant developments.

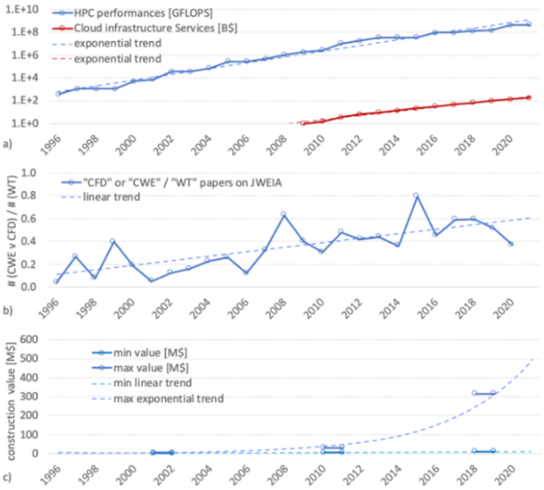


Fig. 1 Time evolution of CWE: (a) Growth of HPC performances and Cloud Infrastructure Services (sources www.top500.org and www.statista.com); (b) scientific production in CWE (source www.sciencedirect.com); (c) construction contract values in which CWE is adopted for the estimation of wind loads on structures (source Optiflow Company)

2. State of the art in CWE practice and codification

As it is well-known CWE results from the convergence of methods and approaches from both Wind Engineering and Computational Fluid Dynamics (CFD). Wind Engineering is best defined as "*the rational treatment of interactions between wind in the atmospheric boundary layer and man and his works on the surface of Earth*" (Cermak 1975). CFD is a branch of fluid mechanics which uses numerical analysis to tackle problems involving fluid flows in a wide range of fields and applications. In the following, the abbreviation "CFD" is used if it is adopted by the author of a cited/quoted study; the acronym "CFD/CWE" is used if reference is made to a tool or to an approach equally shared between CFD and CWE; the abbreviation "CWE" is used if reference is made to an issue that is specific to Computational Wind Engineering only. Recurrent problems which can be addressed using CWE can be generally categorised in environmental applications and structural applications (Potsis *et al*. 2023). The present paper is exclusively devoted to the latter and, namely, to the codified assessment of wind loads to be used for the safe design of civil structures by means of CWE. The date of birth of CWE as a research field is debated (Blocken 2014). For sure, the first International Symposium on Computational Wind Engineering took place in Tokyo in 1992 thanks to the initiative of S. Murakami and co-workers (Murakami 1993, Potsis *et al*. 2023). Ever since then, "*CWE has undergone a successful transition from an emerging field into an increasingly established field in wind engineering research*" (Blocken 2014). Such a transition was made possible by the exponential growth of High-Performance Computing facilities, as shown in Fig. 1(a) in terms of billions of floating-point operations per second (GFLOPS), and resulted in a corresponding growth of scientific research. Such a growth is quantitatively testified in Fig. 1(b) by the trend of the ratio between the number of papers devoted to CWE and to Wind Tunnel (WT) tests, published on the Journal of Wind Engineering and Industrial Aerodynamics (JWEIA). In the Nineties it was nearly nil. Today, it is around 0.6. If this trend holds, CWE will reach quantitative parity in scientific production in about twenty years. On the one hand, the common monotonic increase of hardware facilities and scientific publications suggests a strong correlation between the two variables. On the other hand, the exponential growth of HPC performances and the linear trend of scientific publications hints to the fact that the availability of computational resources is a *necessary but not sufficient condition* to obtain valuable computational results.

2.1 CWE practice in structural design

Data about the use of CWE in design practice are unpublished and difficult to access. To our best knowledge, the first European consulting company specializing in CWE was established in 1998 in France. Fig. 1(c) shows the trend of the range of construction contract values such a company consulted on in its 20+ year-long activity in the field. Three general remarks can be tentatively drawn from this specific reference. First, it confirms CWE entered the design practice and has evolved with a delay with respect to research. Second, at the early stage CWE was a solution for small budget projects, unable to deal with the costs of WT tests. Nowadays, CWE is becoming attractive for large projects as well. Third, the exponential growth of HPC resources has made it possible to deal with more and more demanding projects, by using larger and *potentially* more accurate computational models.

In the authors' opinion, the evidences and remarks above open the door to a conceptual issue, i.e. the relationship between scientific advances and design applications in a growing field as CWE, with special reference to their characteristics time scales of development. In particular, the CWE transfer to practitioners has been understandably approached with caution in the Wind Engineering scientific community, sometimes with skepticism. In 1996 Simiu and Scanlan, pioneers in Wind Engineering, considered CWE unable to face practical design problems: "*For structural engineering* *purposes*, *owing to the computational problems arising in large Reynolds number*, *turbulent*, *separated flows*, [CWE] *current methods are inadequate and/or prohibitively expensive*" (Simiu and Scanlan 1996). In the light of the current state of the art, such a blunt assessment has suffered a premature and quick aging. Five years later, another key figure in the field is less sharp and more open, but appropriately cautious: "[…] *there seems to be an ever-increasing confidence in the results obtained by CFD codes and more and more papers propagate the idea that the numerical wind tunnel does exist today and produces results ready to be used by practitioners*. *In the author's opinion this is at best premature and at worst dangerous with the exception of very limited cases* […] *In spite of some interesting and visually impressive results produced with Computational Wind Engineering*, *the numerical wind tunnel is still virtual rather than real*. *Its potential however*, *is extremely high and its progress should be monitored carefully"* (Stathopoulos 2002).

In the meantime, software development has given a further impulse in popularizing CWE in the market of engineering consulting: (i) the increasing availability of cloud computing, which was firstly coined in 1996 (Regalado 2011) and boomed since 2009 (Fig. 1(a), Cloud Infrastructure Services in billion dollars), has made HPC free from physically owned in-house hardware resources; (ii) friendly Graphical User Interfaces have been developed for most CFD/CWE codes; (iii) integrated working environments, coupling CAD tools with CFD codes, have been developed and are heading toward pre- and post-processing automation (Dorney 2003). Such developments, introduced first in the aerospace industry, have recently landed on the CWE market: nowadays it is not uncommon to come across advertisings of codes devoted to CWE promising a quick and easy modelling of the wind within seconds of starting the application, while others invite to just drag and drop the 3D drawing to receive a report with aerodynamic properties and flow visualizations obtained from a completely automated online solution. Others even argue that Civil Engineers and Architects will master the use of specific CFD software with no required previous CFD experience. Similar approaches to CWE practice dramatically collide with the beliefs of renewed and experienced scholars in the field: "*Especially in the commercial CFD arena*, *user expectations are often that the purchase and use of a 'really good code' will remove from the user the obligation of 'doing his homework*" (Roache, 1997), "*inevitably*, *high quality CFD is often time consuming and costly*. *The validity of the level of expertise required and the time (cost) involved should be carefully evaluated on the basis of its purposes by comparing them with those of other assessment methods*" (Tominaga & Stathopoulos 2013), and "*The use of CFD by designers is 'dangerous and questionable' without 'any supervision by wind engineering or CFD experts*" (Tamura and Phuc 2014).

In short, according to the authors, a twofold critical condition arises. On the one hand, the CWE quick evolution makes it difficult to set permanent limits to its applicability. On the other hand, even the conspicuous consolidated scientific and technical know-how of CWE risks to be overturned and overtaken by the current headlong rush of an unregulated CWE market.

2.2 CWE in guidelines and codes

The issue raised in the previous section is clearly relevant not only to design practice, but also to CWE codification. We tentatively identify three main approaches to CWE codification among the past and present attempts.

In the first approach, standards and codes simply do not mention CFD/CWE. For instance, the superseded ASCE 7-10 (ASCE 7-10 2010) specified that "*permitted procedures*" for wind action evaluation were the calculation procedures provided by the code itself and WT tests, while no mention to CFD was done. Conversely, an entire chapter of the same standard is devoted to specifications for WT procedures. Analogously, other codes do not address the issue of using CFD/CWE for structural design: NTC-2018 in force in Italy does not cite at all CFD/CWE; the Eurocode currently in force (EN 1991-1-4 2005, clause 1.5) generically states that "*properly validated numerical methods may be used to obtain load and response information*, *using appropriate models of the structure and of the natural wind*" and refers to National Annexes for further guidance; the superseded CNR-DT 207/2008 advises the use of CFD for determining wind conditions at construction site only, while no reference is made to its use for the determination of wind actions. It has to be noted that, much more recently, an analogous but more explicit distinction between studies for wind conditions and structural loads evaluation is given in the position paper issued by the UK Wind Engineering Society (Cammelli *et al*. 2022).

In keeping CWE aside, codes and standards have been probably inspired by prudence, to avoid legitimation and popularization of a not yet established approach. Nevertheless, the opposite effect took place in design practice: let us call it the *"CWE wild west"*. It lasted until 15 years ago. It was superseded by a kind of *"law and order"* approach in 2005 and 2009, when two standards set stringent rules about CWE, with different and, say, partially contradictory outcomes. On the one hand, the ISO 4354:2009 standard explicitly prohibits the use of CFD for the wind-resistant design: the standard identifies CFD as a promising tool, but "*with the current state of development of CFD techniques*, *such methods are not able to fully reproduce the fluctuating flow characteristics required to obtain the appropriate fractile of the extreme value distribution of pressure coefficients*, *or the correct correlations between fluctuating pressure coefficients over the surface to give large area (or global) force or moment coefficients*". For these reasons the standard does not recommend the use of CFD for force and pressure coefficient determination. It states in any case that if CFD is used to the scope, its applications must obey to the same requirements specified for the WT practice, in terms of accurate wind environment reproduction and result analysis techniques. On the other hand, the guidelines issued by the Architectural Institute of Japan (AIJ-GWL 2005, Tamura *et al* 2008) encourage the use of numerical approaches: "*CFD* *can predict wind flows around buildings and structures under conditions very close to the actual state*, *and can therefore be used for almost part of wind-resistant design*". The guidelines however set stringent prescriptions on the computational models to be used in terms of proper turbulence modelling and computational parameters, and provide a number of CFD validation results. An analogous approach is adopted by the French National Annex to Eurocode 1 (NF EN 1991-1-4/NA:2008-03, in French): it describes both WT tests and CWE as methods to estimate the wind-induced loads on structures, and critically discusses which modelling approaches are suitable or not for the computational simulation of averaged and peak loads. More recently, the second modification of the Russian Code (Mod2-SP20.13330.2016 2019, Belostotsky *et al* 2019) legitimates the use of CFD without any specific instructions for use : "*for structures with increased level of responsibility* […] *aerodynamic coefficients are* […] *based on the results of 1) physical (experimental) modelling - tests in wind tunnels (appendices G and I); 2) mathematical (numerical) modelling of wind aerodynamics based on numerical schemes for solution of three-dimensional equations of motion of liquid and gas with adequate turbulence models implemented in modern advanced verified licensed software systems of computational fluid dynamics*".

In the Authors' opinion, both the codification approaches reviewed in subsection 2.2 have drawbacks. On the one hand, the lack of codification is a dangerous open door to unregulated market players, as testified in subsection 2.1. On the other hand, a prescriptive, incremental, uneven approach to CWE codification grows old too fast due to its rapid evolution. In other terms, the characteristic time of codification is too long compared to the CWE evolution time, intended both as computational resources availability and scientific progress. Remarkably, an analogous issue was raised by Ballio and Solari (1988) three years after the issue of the born-old Italian Recommendations about the action of wind on constructions (CNR 10012/85). They remarked that "*This development* [ed: of the Wind Tunnel tests] *has called for a rapid bringing into line at the standards level: the leading nations in this sector already consider the drawing up of specific regulations* [ed: reference to ASCE 1985]".

In the light of the synthetic observation above, the ANIV-CWE group decided first to undertake a third path, a *performance-based*, *informative* approach, that grounded in 2018 the new Annex T of CNR-DT R1-207/2018, and recently the Annex K in PrEN 1991-1-4:2021. These *informative* annexes are generally intended to secure a correct and timely technology transfer between the CWE scientific community and the structural designers. In other words, such documents have been primarily conceived in order to (i) allow structural designers to be aware of the fundamental principles of CWE; (ii) codify performance-based good practices for the use of CWE; (iii) warn designers against common misuses of CWE.

Even more recently, the update of ASCE 7 currently in force (ASCE 7-22 2022) has echoed the proposed performance-based approach of Annex T-CNR-DT R1-207/2018. Indeed, in chapter 31 "Wind Tunnel Procedure" permission is given to use results from numerical wind tunnel procedures (i.e., the use of CWE), on the condition that such procedures are verified and validated with respect to a physical wind tunnel test, and subjected to an independent peer review.

3. Fundamental principles for CWE codification

In the following, the main underlying concepts that ground the Annex T in CNR-DT R1-207/2018 and K in PrEN 1991-1-4:2021 are listed and briefly commented.

P1. *CWE doesn't match CFD*. Computational Wind Engineering is the discipline that applies Computational Fluid Dynamics to either environmental or structural Wind Engineering, bringing together concepts and methods of both fields. Consequently, CWE does not coincide with CFD; indeed, nearly 25 years ago Murakami (1998) already emphasized some of the peculiarities and specific difficulties of CWE. In this regard, it is crucial to stress that a CFD expert is not necessarily a CWE expert too; therefore, specific guidelines for CWE are necessary.

P2. *A universal CWE model does not exist*. Any computational model is approximated, based on validity assumptions that are suitable for some problems but are not for others. Every model should reproduce with proper accuracy all the significant features of the physical phenomenon relevant to the application of interest. It follows that the label "CWE" refers to an approach which needs proper case-by-case specification and it is no way assimilable to a universal and univocal tool able to predict wind loads thanks to the so-called "virtual wind tunnel" as intended in commercials.

P3. *Setting the class of problem first*. A direct consequence of the point above is that the identification of the *class of problem* which the specific application belongs to is pivotal, as emphasized for instance by Castro and Graham (1999): "*without a sound understanding of the fluid mechanics appropriate to the particular problem being attacked*, […] *great caution is required in using CFD as an integral part of the design process*". The *class of problem* results from the *a priori* qualitative prediction of: (i) the incoming wind features at the project site; (ii) the aerodynamic behavior of the structure; (iii) the structural type; (iv) the focus of the analysis; (v) the design stage to which the analysis relates. The concept of *class of problem* is further discussed in Section 5.2.

P4. *Validate first*. A fundamental principle of most codes and standards about CWE is that any computational model must be validated against experiments or high-fidelity computational simulations selected from scientifically and technically qualified sources. This goes against "*the concept of a numerical wind tunnel generating quantitatively meaningful design data without careful case-related experimental validation*", about which Leschziner (1990) was already doubting "*it is at all a sensible objective to pursue*". Validation plays a crucial role, and tightly relates to the *class of problem*. Indeed, the *validation of a numerical model is mandatory for the class of problem to* *which the application belongs to*, but it is not necessarily requested for every single application. On the other hand, the use of a verified code (see, e.g., Roache 1997, and Oberkampf *et al* 2004) is a necessary but not a sufficient condition for the reliability of a computational analysis of an engineering problem. The validation requirements in Annex T of CNR-DT R1-207/2018 and Annex K of PrEN 1991-1-4:2021 will be discussed in Section 5.2.

P5. *Designer*, *CWE Specialist*, *Skilled Controller*. Interdisciplinary scientific and technical skills in both CFD and Wind Engineering, as well as long term professional experience are required to be in charge of CWE simulations. This is necessary to guarantee the control of the setup conditions, computational model and results obtained, analogously to what is expected for the manager of WT tests. The mere availability of CFD/CWE software and being familiar with its user-interface do not qualify for carrying out the task. The person who fulfills the requirements above is called *CWE Specialist*. Usually, the structural *Designer* does not have the specific competences required to play the role of *CWE Specialist*. The informative Annexes in CNR-DT R1-207/2018 and PrEN 1991-1-4:2021 neither aspire nor suffice to train a *CWE Specialist*. As already declared, the Annexes above are intended to allow the structural *Designer* (or, more in general, the final customer of a CWE study) to be aware of operational procedures for the most recurrent CWE simulations and to dialogue and interact in an informed manner with the *CWE Specialist*. Finally, another *CWE Specialist* without conflict of interest is recommended to be entrusted by the customer to act as *Skilled Controller* in order to perform an independent control in case of particularly demanding design cases (see Section 5.3).

4. Computational model

Central in a CWE study is the setup of the computational model. In the following, the structure adopted for its description in Annex T of CNR-DT R1-207/2018 and Annex K of PrEN 1991-1-4:2021 is summarized.

For the purpose of exposition, it is useful to consider the computational model as the combination of numerous interacting components, whose efficacy can be evaluated only by considering the model behavior as a whole. Each of the adopted components inevitably introduces errors, generally defined as the difference between the results obtained by the component application and a reference exact solution.

It is useful to subdivide components into two categories: the *modelling approach* and the *numerical approach*. The first one collects components which directly impact the physical phenomena that the simulation will/will not be able to reproduce and are, essentially, modelling choices corresponding to hypotheses and approximations introduced in order to build the model.

The second one collects approximations needed to discretize and solve the equations which govern the model behavior, e.g., the equation describing the fluid flow (Hirsch 2007, Moukalled *et al*. 2016, Versteeg *et al*. 2007) and, possibly, the structural motion. A schematic and non-exhaustive view of the main components belonging to each of the aforementioned categories is shown in Fig. 2.

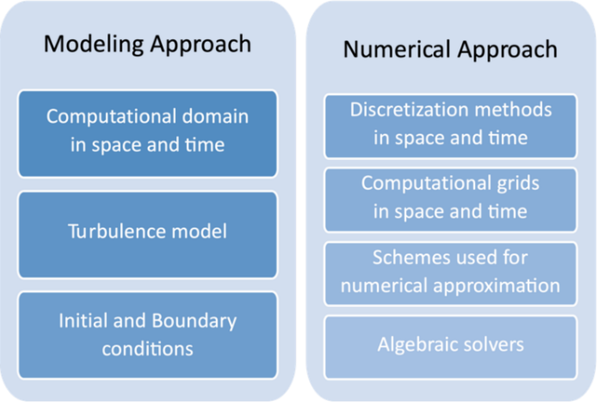


Fig. 2 Scheme of the components of a CFD/CWE model

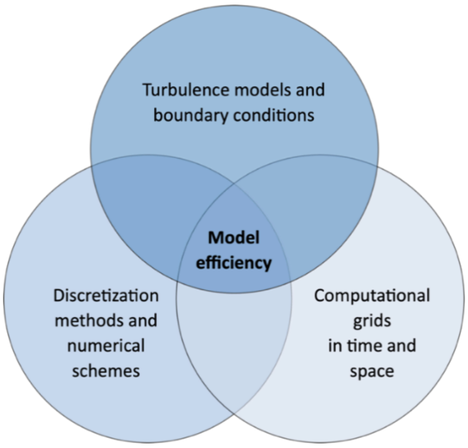
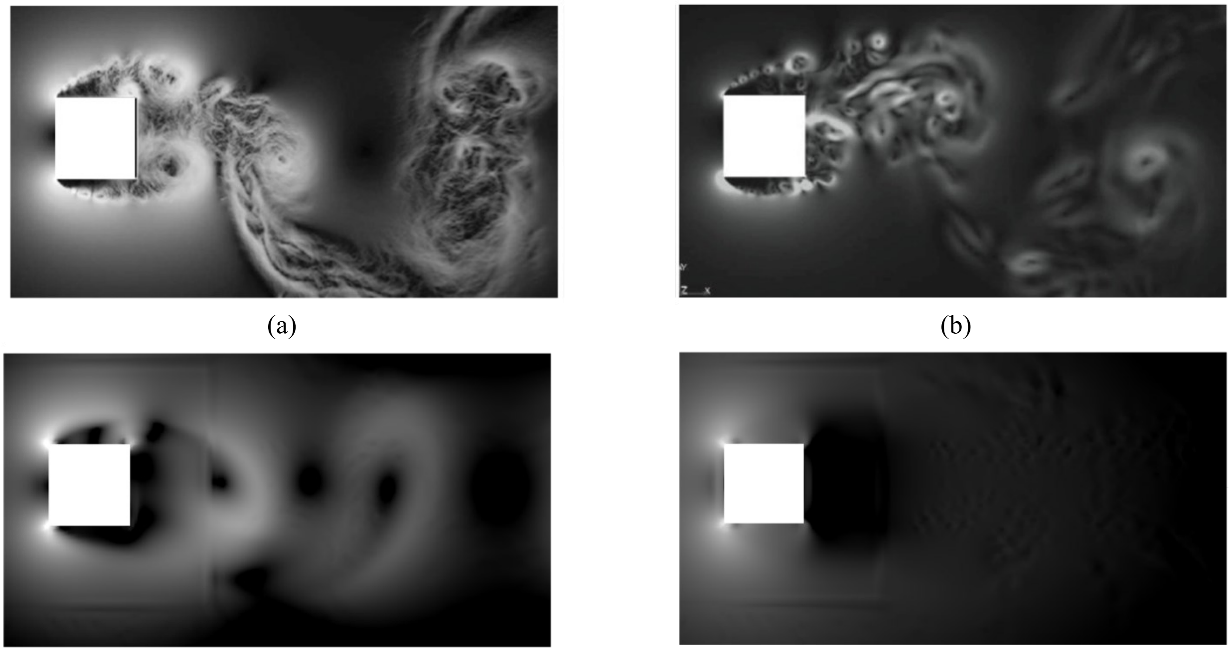


Fig. 3 Scheme of the main components and their interrelations affecting the overall model efficiency

In essence, setting up a computational model requires to combine components from both the *modelling* and *numerical* approach, aiming at optimizing the model efficiency, intended as the ratio between the accuracy of the obtained results and the computational costs. A schematic overview of the main components is sketched in Fig. 3, highlighting that the model efficiency arises from interrelations rather than from a chain of sequential choices.

It shall be noticed again that, as no universal optimal model exists for CWE, such optimization must be performed considering the *class of problem* to which the case under consideration belongs. In fact, within the same computational model, the accuracy of different quantities can greatly vary, ranging from very high to extremely poor. A typical example is represented by global along-wind forces, which are often well-described also by poorly performing models and relatively insensitive to the components adopted for the *numerical approach*. Conversely, the local peak pressures distribution useful for cladding design is usually characterized by low accuracy if


(a) (b)

(c) (d)

Fig. 4 Vortex shedding past a square cylinder: instantaneous magnitude of the pressure gradient simulated by (a) DNS (after Trias *et al*. 2015); (b) LES (after Cao and Tamura 2016); (c) URANS k-ε RNG; (d) URANS k-ε STD

inadequate models are used, and might prove sensitive to components of both the *modelling* and *numerical approach* (see, for instance, Xing *et al*. 2022).

Unfortunately, as already stated, the accuracy of the simulation depends not only on the selected components themselves, so to say singularly taken, but also, to a large extent, on their interrelation and compatibility.

A single guiding principle is suggested for the model specification: each component shall be selected aiming at evening up the introduced errors with respect to all the others components, in such a way that the overall committed error is compatible with the *class of problem* under investigation.

It must be noticed that such guiding principle encapsulates the ontological interdisciplinary nature of CWE, in which the *modelling approach* and *the numerical approach* represent the roots of the discipline into CFD, while the *class of problem* represents its branches extending toward Wind Engineering.

Summarising, discrepancies between different models and between models and reality shall be considered as unavoidable in CWE applications, and controlling and managing such uncertainties considering their effect with respect to applications is a matter exclusive to CWE, which cannot be directly inferred by the knowledge of CFD and Wind Engineering singularly considered.

Without aiming at a thorough treatment, a brief and not exhaustive description of some of the most important components contributing to the definition of the *modelling* and *numerical approach* is provided below.

4.1 Turbulence models and boundary conditions

Turbulence and its modelling surely represents a critical point for any CWE study, and it is probably the aspect which disorients the most non-specialists. It is an aspect of the analyses to which all choices are tightly related to. The *turbulence model*, i.e., the component of the *modelling approach* used to represent the effects of turbulence, is characterized by an extremely wide range of options which differ between each other in qualitative terms.

Firstly, it is important to highlight that the majority of structures relevant for CWE applications are immersed in turbulent flows (Holmes *et al*. 1990), in which velocity and pressure randomly fluctuate in space and time.

Such fluctuations, which can be though as caused by the presence of eddies, are known to deeply affect the flow field organization in the proximity of immersed bodies (e.g., Bearman and Morel 1983, Noda *et al*. 2003). Such eddies are characterized by a wide range of characteristic dimensions, denoted as *scales*. For CWE applications, the largest ones have *scale* in the order of 100-200 m (Solari 1993), while the *scale* of the smallest ones is dictated by the mechanism governing energy dissipation into heat (less than 1 mm). Such subdivision in *large scales* and *small scales* is conventional and qualitative in nature and, in reality, a continuous cascading of *large scales* into *small scales* is observed (Pope 2000).

Navier-Stokes equations are able to fully represent all the scales present in a turbulent flow and their interaction.

When numerical simulations are performed aiming at simulating all scales they are called Direct Numerical Simulations (DNS).

As it is well-known, explicitly modelling all *scales* is impossible in the design practice, due to the unaffordable computational burden deriving from the high resolution (in space and time) needed to resolve the *small scales*. Unfortunately, the effect of the *small scales* on the large

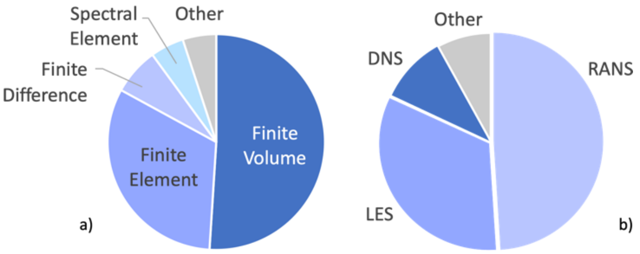


Fig. 5 Discretization methods of the Navier Stokes equations (a) and turbulence approaches (b). Statistics over the most common academic and commercial CFD/CWE codes (source https://www.cfd-online.com/Wiki/Codes, ensemble cardinality: 132 codes)

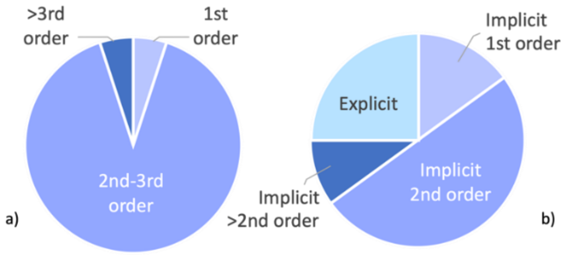


Fig. 6 Spatial (a) and temporal (b) discretization schemes. Statistics over the most common academic and commercial CFD codes (source https://www.cfd-online.com/Wiki/Codes, ensemble cardinality: 132 codes)

ones is not negligible, so that their effect must be approximated using a *turbulence model* (Wilcox 1998). Such an approximation usually consists in the introduction of the so-called flow “turbulent viscosity” in addition to fluid viscosity, accounting for the mixing effect operated by turbulence.

Turbulence models can be broadly subdivided into two categories: *scale-resolving* and *non scale-resolving*. *Scale-resolving* models aim at explicitly simulating scales which can be well-resolved by the adopted *time and space grid* (see Section 4.3) and approximate the effect of smaller ones, usually by means of the aforementioned turbulent viscosity. A well-known and widely adopted typology of models falling into this category is represented by Large Eddy Simulation (LES). Conversely, *non scale-resolving* models aim at simulating explicitly only the mean flow, which can also be non-stationary. In such a case the mean shall be intended in the Favre-averaging sense. Such models aim at approximating the effect of all scales of turbulence on the mean flow. Well-known and widely adopted typologies of models falling into this category are represented by steady Reynolds-Averaged-Navier-Stokes (RANS) and their Unsteady counterpart (URANS). The flow around an elongated prism of square cross-section simulated using various *turbulence models* is reported in Fig. 4, in which the qualitative difference between DNS (Fig. 4(a)), *scale-resolving* (Fig. 4b) and *non scale-resolving models* (Fig. 4(c) and 4(d)), can be easily observed. Notice that the standard k-ε model (URANS k-ε STD) predicts a steady flow, while the real flow is clearly unsteady, as captured by the other models.

Finally, we mention the existence of hybrid models, characterized by intermediate behaviours between the two aforementioned main categories (Menter 2010), usually conceived in order to address particular shortcomings of either of them, targeting particular applications.

It is also worth mentioning that, due to the extremely high computational requirements involved by the turbulence modelling in the proximity of solid walls, *ad hoc* complementary models are locally used both in *scale-resolving* and *non scale-resolving* models (e.g., wall-functions or damping-functions approaches).

The *turbulence model* choice must be adequate for the *class of problem* under investigation and poses severe restrictions to the choice of all the other model components in order to ensure their mutual compatibility.

Unfortunately, none of the available *turbulence models* can be considered truly superior to all the others, as all of them represent a different balance between accuracy of the simulation and computational cost, suitable for particular *classes of problem*. Notice also that results must be expected to vary considerably, even when using models falling into the same category, often showing qualitatively different predictions.

Another aspect which deserves careful consideration is the definition of inflow boundary conditions (b.c.). On such regard, it must be reminded that civil structures are usually immersed in the Atmospheric Boundary Layer (ABL), which is characterized by variations of the mean velocity and turbulence properties along the vertical direction. Depending on the adopted *turbulence model*, the specification of the incoming turbulence can change considerably. In the case of *non scale-resolving* models, the boundary conditions for the turbulent variables must be assigned based on the vertical profiles for the turbulence-related bulk quantities (e.g., turbulence intensity and integral length scale, Richards, 2019). In the case of *scale-resolving* models, unsteady random fields of the velocity fluctuations expected on site due to turbulence must be explicitly generated. Various techniques are available to this purpose, which can be mainly classified into the categories of “precursor simulations/recycling methods” and “synthetic generation” (Wu 2017, Thordal *et al*. 2019). Both categories are applicable to CWE and might be used to generate appropriate turbulent inflow conditions.

4.2 Discretization methods and numerical schemes

As it is well-known, the exact solution of the (integro)-differential equations governing the wind flow around a structure cannot be analytically obtained.

It is therefore necessary to discretize the problem to obtain an approximation in terms of algebraic equations that can be solved numerically by using a computational approach. CFD codes can be based on different *discretization methods* of the Navier Stokes equations (without considering here Lattice-Boltzmann approaches): we can mention the Finite Volume Method, the Finite Element Method, the Finite Difference Method (e.g., Peyret and Taylor 1983), and the Spectral Methods (e.g., Canuto *et*

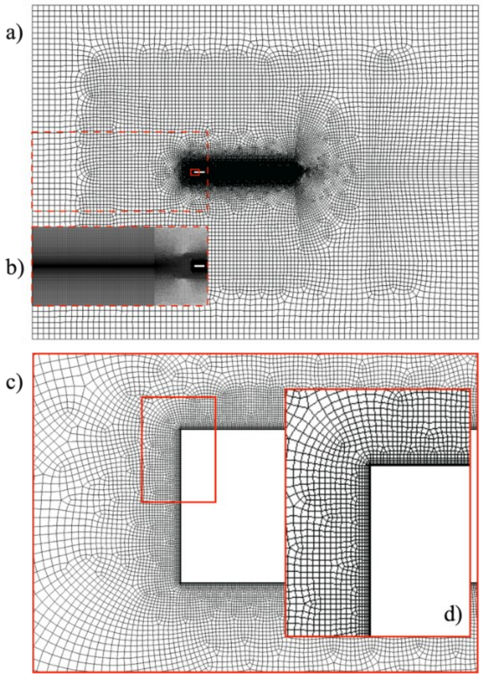


Fig. 7 Computational grid around a rectangular cylinder in the transverse plane: whole computational domain (a) without or (b) with incoming turbulence; (c) close-up view around the leading edges; (d) further zoom around the upper leading edge. Fig. (a), (c), (d) after Bruno *et al.* (2010)

al. 1988). For CWE applications, the Finite Volume Method is widely used and implemented in most of the numerical codes currently available for industrial simulations, as shown from the statistics in Fig. 5(a), where the discretization methods of the Navier Stokes equations used by the most common academic and commercial CFD/CWE codes are summarized. As can be seen in Fig. 5(a), more than 50% of the considered codes adopt a Finite Volume Method. The Finite Element Method is also widely used, whereas Spectral Elements and Finite Difference methods have at the state of the art a limited diffusion. As for turbulence approaches discussed in Section 4.1, RANS is the most common approach (approximately one-half of the considered codes), followed by LES and DNS (Fig. 5(b)).

In order to translate the (integro)-differential operators of the Navier-Stokes equations into an algebraic form, numerical approximation schemes are used. Such schemes introduce a discretization error (Ferziger and Peric 1996), understood as the difference between the exact solution of the aforementioned equations and its algebraic counterpart. The main CFD/CWE codes offer several alternative schemes, the accuracy of which varies with the geometry of the cells, the type of grid (structured or unstructured, uniform or non-uniform), and the dimensionality of the space domain.

Different approximation schemes can be employed, and the choice among them has to be performed according to the characteristics of the problem under consideration. In general, it is advisable to use schemes characterized by *second-order accuracy* or higher, with regard to both spatial and temporal derivatives, and indeed these are usually adopted by codes (see Fig. 6). Indeed, it is found that more than 90% of the codes have second-order-accurate spatial discretization schemes (or, sometimes, third-order), and the second-order accuracy is also the most common choice when dealing with the implicit time-advancing scheme, even if the explicit scheme is also available in about one-quarter of the codes.

It is worth mentioning that the introduction of the discretization error can also be seen as a fictitious modification of the underlaying governing equations. Such modification, depending on the adopted schemes, can lead to the introduction of fictitious dispersion or diffusion terms, which are of particular importance for CWE computations and often denoted as “numerical viscosity”. In particular, attention shall be paid to the discretization of convective terms in the balance equations, aiming to obtain a good compromise between low numerical viscosity and solution stability.

4.3 Computational grids in time and space

The Finite Volume Method, as well as most alternative methods, requires the discretization of spatial and time domains. The spatial domain is discretized through a finite number of elementary sub-domains commonly called cells. The time domain is discretized using a finite number of times separated by elementary time intervals commonly called time steps. The set of cells defines the grid in space (namely, the mesh), while the union of the grid in space and of the time discretization defines the computational grid. The grid determines the spatial and temporal resolution of the computational simulation.

The choice of the computational grid is a crucial step in the context of CFD/CWE simulations, and naturally associated with the three- or two-dimensional geometry of the spatial domain (see subsection 5.2). As a matter of fact, such a choice strongly affects the main properties of the computational solution for Wind Engineering applications in terms of stability, accuracy, and computational costs.

The computational grid shall be generated to ensure sufficient spatial and temporal resolution and to reproduce the turbulence scales that, in accordance with the turbulence model adopted, are relevant for the phenomenon under investigation. In addition, grid requirements introduced by particular components of the computational model shall be accounted for (e.g., requirements involved by wall-functions or damping-functions).

Particular attention shall be paid to the discretization of the regions of the computational domain where the flow is expected to be characterized by large gradients of the flow variables. In particular, the discretization error tends to increase when large gradients are present, so that evening them up requires to increase the grid density in such zones, according to the general principle reported in Section 4. For instance, mesh refinements in the boundary layer region near solid walls, close to possible separation points (e.g., near edges) and in the wakes have to be properly considered. We notice that, while in the first case the directionality of the expected flow variations conveniently

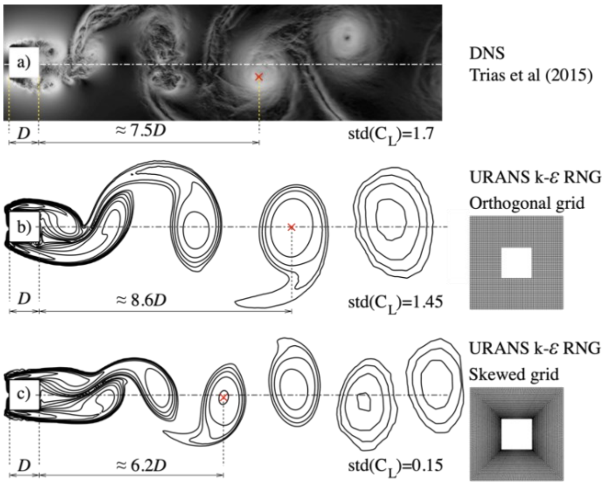


Fig. 8 Effects of grid quality on the instantaneous wind flow and around a square cylinder and on the induced lift coefficient: (a) reference solution (DNS after Trias *et al.* 2015), and (b) the one obtained with an orthogonal and (c) a skewed grid (URANS k- ε RNG after Bruno and Oberto 2022)

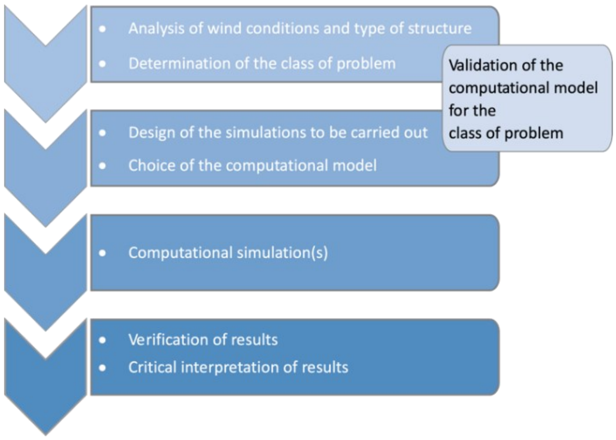


Fig. 9 Recommended schematic workflow of a CWE study

allows using stretched grids, leading to boundary layer cells, isotropic refinements are more suitable for the latter case (Spalart 2001). As an example, Fig. 7 shows a grid with an adequate discretization of the mentioned zones and with high cell quality. In the considered case, a body-fitted, structured grid layer is generated at the wall, with a nearly constant grid spacing normal to the cylinder wall able to correctly resolve wall turbulence (Fig. 7(d)). Then, an unstructured quadrilateral grid is used in the remaining part of the transverse plane of the cylinder to obtain an effective cell distribution on the basis of the expected flow phenomena to be simulated (Fig. 7(c)). Upstream of the investigated body, the computational grid should be fine enough to accurately transport turbulence from the inflow boundary to the object of the study, without numerical effects. In case of smooth incoming flow, a grid as in Fig. 7(a) is fine enough; on the contrary, in case of *scale-resolving* turbulence models and turbulent incoming flow a grid as in Fig. 7(b) is necessary.

The computational grid shall also be realized to guarantee cells with sufficient geometrical quality, quantified by appropriate metrics, to avoid loss in simulation accuracy and stability. In particular, it is good practice to avoid as much as possible distorted and/or highly stretched cells. Indeed, possible sources of errors are the grid refinement/coarsening rate, the lack of cell orthogonality, and the cell skewness, which is defined as the distance between the midpoint of the cell face and the intersection between the same face and the segment linking the adjacent cell centers.

An example of the treacherous effects of localized poorly shaped (skewed) cells around a bluff body is given in Fig. 8. In spite of an acceptable qualitative description of the vortex shedding, the lift force fluctuations are dramatically and unsafely underestimated by a factor 10: “*the greatest disaster one can encounter in computation [...] are results that are simultaneously good enough to be believable but bad enough to cause trouble*” (Ferziger 1993).

We finally remark that one of the workhorses of numerical analysis for the assessment of the numerical solution quality, i.e., grid independence study, sometimes appears to be of limited help. First, the computational costs required to rigorously perform global multi-levels, systematic and uniform mesh refinements over the whole domain are usually prohibitive in 3D domains. Indeed, a single global mesh refinement level in 3D increases the number of volume elements by a factor of eight, the cost of the solution increases with the square of the number of degrees of freedom (Thompson and Thompson 2017) and the grid independence study becomes practically intractable. Thus, systematic but local mesh refinement is usually more economical and as effective as global mesh refinement. Second, such refinements can conflict with the mesh requirements involved by turbulence treatments at solid walls (e.g., wall-functions). Overall, solution verification is necessary before model validation (Oberkampf and Trucano 2002). In other words, proving a solution to be mesh independent is an important indication for the CWE Specialist, but should never be used for the purpose of validation. From this point of view, in order to avoid misconceptions, it would probably be more appropriate to state that the CWE Specialist is required to ensure that results of interest do not significantly vary with the mesh size, rather than requiring a mesh independence which, literally taken, is extremely difficult to be reached and even more difficult to be correctly assessed.

5. CWE instruction for use

In the previous sections the main CWE principles are set out in general terms (Section 3), and the components of the computational model are discussed (Section 4). In the following, some key passages of a typical CWE study are detailed in term of required performances and exemplified with respect to real world design cases. In particular, the workflow recommended within a CWE study is schematized in Fig. 9. In the following, its main steps are discussed and examples considering real world application are given.

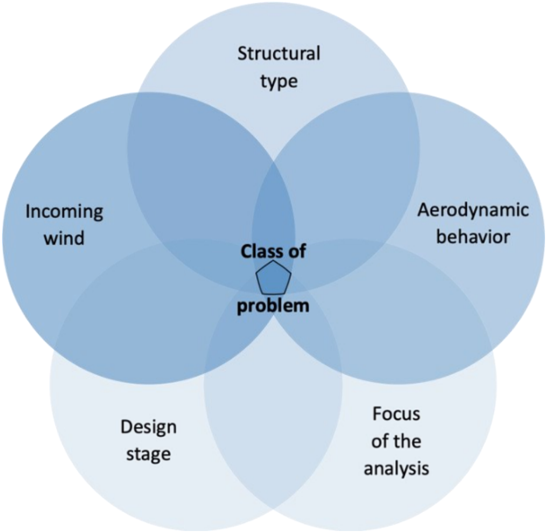


Fig. 10 Key features of the design case that define the class of problem to which it belongs to

5.1 Recognizing the class of problem

As already emphasized, the very first and challenging task towards effective CWE applications consists in *recognizing the class of problem* to which the specific design case belongs to (first block in Fig. 9). To this aim, the *CWE Specialist* shall (Fig. 10):

- check the expected incoming wind conditions at the project site (e.g., low or high turbulent wind, quasi-uniform or varying velocity profile in the vertical direction);
- qualitatively predict the aerodynamic behavior of the structure (e.g., attached, fully separated, separated-and-reattached boundary layer flow);
- recognize the structural type (e.g., beam-like, spatial structure, suspended, trussed) and material (e.g., reinforced concrete, steel, wood);
- identify the focus of the analysis (e.g., wind-induced Ultimate or Serviceability Limit State, static-equivalent, dynamic or aeroelastic structural response to the wind, global or local effects);
- clarify the design stage to which the CWE simulation shall contribute to, and its objective(s) (e.g., conceptual, preliminary, detailed, as-built design).

The key features above can be provided by the structural *Designer*, by specialists other than the CWE one, or set by the *CWE Specialist* him/herself. It clearly appears that such a fundamental preliminary task, analogously adopted to establish the most appropriate setup in WT tests, requires specific skills and experience, and it cannot be faced through automatic and univocal processes, nor via fully automated computational tools. For purely illustrative purposes, four *classes of problem* determinations are exemplified in Table 1, sorted by increasing level of difficulty from the CWE standpoint.

5.2 Planning the simulations and selecting the model components

Once the *class of problem* is determined, the set of simulations shall be planned and the computational model shall be defined by selecting the most suitable components (second block in Fig. 9). The model components shall be chosen consistently with the *class of problem* and in order to secure the compatibility among them, according to the main principle reported in Section 3.

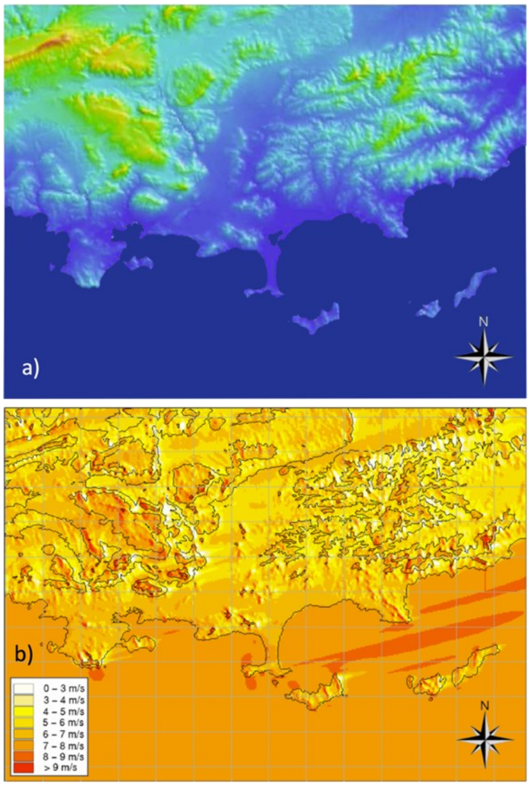


Fig. 11 Computational simulation of the influence of the orography on the wind speed distribution by a non scale-resolving RANS Realizable k-ε model: (a) digital orographic model; (b) simulated flow field (mean velocity at 10 m height). Credits: Optiflow Company

Before proceeding, it is important to notice that: i) some class of problem might be particularly delicate and/or difficult to be analysed using CWE; ii) a component necessary for a specific class of problem might simply be not available in specific software. The expectation that software dedicated to CWE shall be able to solve all *classes of problem* might lead to a perspective bias, which might induce unexperienced users to select the most reasonable between the available choices within an incomplete set. If this happens the model is not simply inaccurate. Rather, it might be totally unable to reproduce relevant physical aspects. This is the case, for example, of aeroelastic interactions, and/or turbulence in the approaching flow, and/or the behaviour of compressible flows, whose modelling is not necessarily available in all software dedicated to CWE.

In the following some relevant recommendations are provided. The adopted exposition order is insignificant and chosen only for the sake of clarity, presenting when possible those having direct analogues in WT tests first, and, then, those more typical of CWE.

- *Size of the spatial domain*. The extent of the

Table 1 Examples of classes of problem in CWE

Class of problem	#1	#2	#3	#4
Incoming wind	Low turbulence, uniform wind	ABL wind	Urban, turbulent ABL	Urban, turbulent ABL
Aerodynamic behavior	Separated-and-reattached flow	Streamlined box girder and bluff arch/barriers	Bluff body, massive boundary layer separation at corners	Bluff body, massive boundary layer separation at corners
Structural type	Bridge deck	Bridge	Tall building	Tall building facade
Design stage	Preliminary design / aerodynamic shape optimization	As built design / wind loads	As built design / Strength assessment vs peak global loads	As built design / Cladding design vs peak local loads
Focus of the analysis	Vortex-induced vibration, aeroelasticity, traffic safety	Global vortex-induced response	Global gust and vortex-induced response	Single cladding panel response to local peak pressures

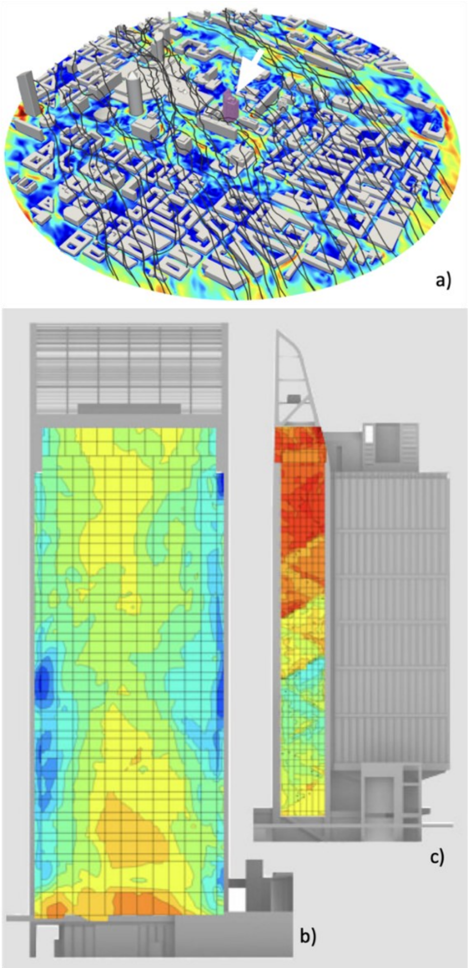


Fig. 12 Buildings surrounding the considered tower (in violet and pointed by a white arrow): (a) streamlines and instantaneous wind speed at pedestrian level in the whole surroundings; (b) instantaneous pressure field on the main facade, (c) envelope of the peak pressure coefficient on two building facades using a scale-resolving LES model for cladding design. Credits: Optiflow Company

computational domain in space shall: (i) avoid improper intrusive effects of the b.c. on the local flow around the structure, and (ii) allow the proper evaluation of the aerodynamic effects of obstacles in its surroundings. The first requirement is analogous to the minimization of the blockage effects in WT tests. Attention should be paid to the distance between the structure and all the domain b.c. Their most effective position might be defined checking the convergence of the results adopting progressively larger domains (see e.g., Abu-Zidan *et al*. 2021). Usually, the maximum blockage ratio is set to 5% in WT tests (e.g., ASCE/SEI 49-12 1997, PrEN 1991-1-4:2021 Annex K), even though some authors deem as acceptable measurements obtained under slightly higher values of the blockage ratio, in any case less than 10% (e.g., Houghtno and Carruthers 1976, Liu 1991, Choi and Kwon 1998). Nevertheless, the maximum values of blockage ratio generally adopted in CWE studies are significantly smaller than 5% (see, for instance, Bruno *et al*, 2014). The second requirement implies that the orography (e.g., in Fig. 11) and adjacent obstacles (e.g., in Fig. 12(a)) shall be explicitly accommodated in the domain. The extent in plan of the so-called “proximity model” shall be large enough to include all the obstacles upwind the structure whose wake significantly affects the structure aerodynamic behavior. In

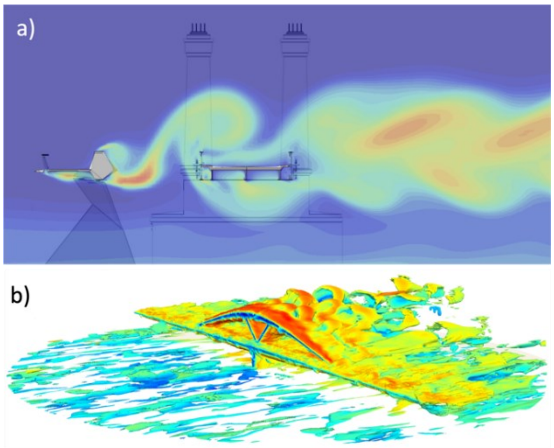


Fig. 13 Typical CWE numerical models for bridges: (a) 2D non scale-resolving sectional URANS k-ω model of a suspended bridge deck in the wake of a footbridge for preliminary design (instantaneous wind velocity); (b) 3D scale-resolving LES model of an arch bridge for detailed design (instantaneous vorticity isosurfaces). Credits: Optiflow Company

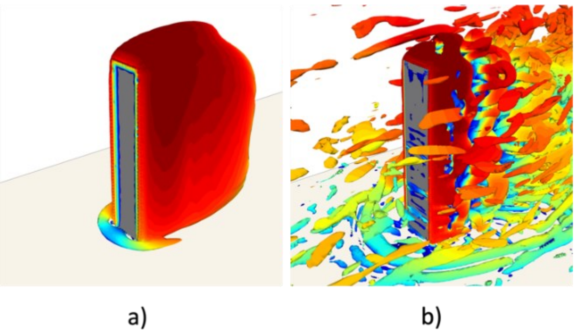


Fig. 14. Flow patterns around a tall building (square base, height-to-base ratio 5) simulated by a *non scale-resolving* URANS k-ω (a) and *scale-resolving* LES (b) turbulence models (instantaneous vortical structures visualized by means of isosurfaces of the second invariant of the velocity gradient, after Fransos and Lo Giudice 2015)

some cases, when the construction of buildings of remarkable size is already planned on the site, surroundings in their present and future configurations might need consideration.

- *Geometry of the spatial domain*. Three-dimensional (3D) spatial domains shall be generally adopted, as it is well-known that turbulent flows are instantaneously always 3D. Nevertheless, in the case of line-like structures (i.e., body obtained by the extrusion of a cross-section along a line), the time-averaged flow may be two-dimensional (2D). In these cases, the adoption of a *scale-resolving* approach and of a 2D computational domain in space is a rare and generally not recommended scenario, that needs to be carefully discussed and verified during the validation and interpretation of the results. For instance, considering sectional models for analyzing the aerodynamics of long-span bridge decks, the assessment of the mean flow features, useful in preliminary design stages, may be addressed with *non scale-resolving* 2D models (e.g., in Fig. 13(a)), especially for streamlined bodies.

On the other hand, for detailed design phases, the use of 3D *scale-resolving* approaches is generally recommended (e.g., in Fig. 13(b)). The use of 2D models shall always be addressed with caution when non-streamlined bodies are considered (Bertani *et al* 2022).

- *Duration of the time domain*. It is necessary to carry out simulations that are sufficiently extended in time in order to ensure that: (i) results are not affected by initial conditions and refers to a fully developed flow; (ii) statistical convergence in time is reached. As regards the first aspect, the initial part of the simulation is usually discarded during the postprocessing. Such discarded time must also consider the time needed for the inflow turbulence, if present, to reach the target building from the inlet patch. As regards the second aspect, it is required that the flow quantities of interest do not significantly depend on the extent of the time window used for their calculation, so that if a longer simulation is used, no significant changes occur. It is important to notice that statistical convergence imposes very different constraints depending on the considered quantity of interest. Generally speaking, convergence is quicker (i.e., the quantity can be evaluated using shorter time windows) for time-averaged values, while the estimation of peak values, e.g., local peak pressures useful for cladding design, require longer simulation times. As a general rule, the higher the considered statistical moment (time-average, standard deviation, skewness, etc…), the higher is the importance of rare events and, thus, the slower is statistical convergence (e.g., Bruno *et al*. 2010). It shall be noticed that reaching statistical convergence is generally more demanding for CWE than WT tests, due to the different cost structure of the analyses. In fact, the former is characterized by a relatively small initial cost and a linear cost increase with the simulation time, while the latter is characterized by a strong initial investment due the model manufacturing and wind-tunnel setup and a relative ease in increasing the study time duration.

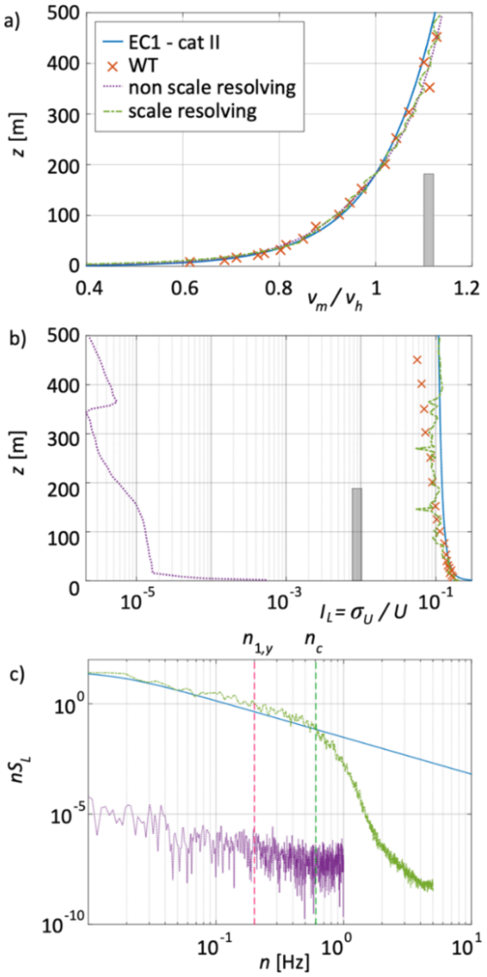


Fig. 15. Incoming wind to a tall building simulated by *non scale-resolving* URANS k-ω and *scale-resolving* LES time-dependent simulations (after Fransos and Lo Giudice 2015): (a) incoming wind mean velocity profile v_m normalized with respect to the velocity at the tower height v_h; (b) profile of the turbulence intensity I_L; (c) spectrum of the longitudinal wind velocity S_L

- *Wind directions*. The number of considered wind directions, i.e., yaw angles α in the horizontal plane (usually for buildings or similar structures), or angles of attack α in the vertical one (usually for bridge decks) shall be enough to account for the on-site conditions, and to include critical values, i.e., angles at which the flow switches from one regime to another, e.g., from attached to separated, or vice versa. Both conditions are often but not always fulfilled adopting $0<\theta<360°$, $\Delta\theta=15°$, or $-10<\alpha<+10°$, $\Delta\alpha=2°$. It can be generally possible to consider a smaller number of wind directions within the preliminary design stage.

- *Compliance with similarity criteria*. In a similar way to WT tests, it is important to ensure aerodynamic similarity between the model and the studied structure. Such an issue (e.g., in-situ and model Reynolds number, Re) is more restrictive and problematic in WT tests than for CWE, because it is possible to computationally simulate full-scale conditions, although this might affect the simulation stability and increase the computational cost.
- *Modelling approach to turbulence*. As already emphasized, the adopted *turbulence model* should be able to provide the results relevant to the considered *class of problem*. An example is provided in Fig. 14 with reference to a tall building: if for optimization purposes the preliminary design is interested in time averaged global forces, a *non scale-resolving* model might be deemed sufficient (Fig. 14(a)); if buffeting forces or local peak pressures are addressed, a *scale-resolving* approach is mandatory to simulate all the relevant flow scales (Fig. 14(b)).
- *Incoming flow features*. The simulated incoming flow shall: (i) reproduce the features of the turbulent wind in the ABL for the ground aerodynamic roughness of the design site for each direction; (ii) be consistent with the adopted turbulence model. The first requirement is analogous to the check of the wind conditions in the test section of an empty WT. More explicitly, the *CWE Specialist* shall check that the wind impinging on the studied structure is representative of the one expected on site, in agreement with the adopted turbulence model.

In other words, the adequacy of the procedures used to represent inflow turbulence shall be evaluated based on the quality and quantity of turbulence actually impinging on the body (Lamberti *et al*. 2020) when orography and surroundings are removed (as in standard WT tests), and not basing on the values set at the inflow boundary, which is usually located far upstream the structure object of the analyses.

For *scale-resolving* simulations, space and time correlations of the incoming turbulence shall be reproduced and, in particular, spectra of the incoming turbulent fluctuations shall be well-represented up to an appropriately estimated cut-off frequency (Patruno and de Miranda, 2020). As an illustrative example, we consider again the tower shown in Fig. 14. Both *scale*- and *non scale-resolving* models accurately predict the time-average velocity profile (Fig. 15(a)). Conversely, dramatic differences between the two modelling approaches appear when the unsteady flow characteristics are inspected, despite both models are time-dependent (URANS k-ω). In particular, the URANS *non scale-resolving* model returns a time-dependent solution of the wind velocity that explicitly includes the fluctuating component of the ensemble-average flow only, but not the turbulent component. That is the reason why the turbulence intensity (Fig. 15(b)) and the spectral density (Fig. 15(c)) are underestimated by five orders of magnitude. Only the *scale-resolving* approach is able to reproduce the level of turbulence intensity (Fig. 15(b)), and the spectrum of the incoming turbulence up to the cut-off frequency (Fig. 15(c)). In general, the cut-off frequency n_c, besides fluid dynamic considerations, shall be greater than the highest frequency of interest for the design case (in the example, the natural frequency of the structure in the crosswind direction $n_{1,y}$, Fig. 15(c)).

- *Numerical approach*. The numerical schemes shall be selected in order to keep the overall numerical viscosity low compared to the one proper of the fluid plus the one introduced by the turbulence model. In this perspective, first order schemes should be generally avoided for flows with high Reynolds number and low turbulence intensity.
- *Computational grid density*. The computational grid density shall be compliant with the turbulence model, the wall treatment, the inlet b.c. and the numerical schemes. Two cases are proposed in the following for illustrative purposes, by referring to different types of turbulence models, structures and focus of the analysis. Both examples are limited to the Finite Volume Method based on second-order accurate schemes.

- Models aiming to analyze the turbulent wind response of tall buildings or long-span roofs with simple geometry and without construction details, adopting a *scale-resolving* turbulence model and a three-dimensional computational domain in space: spatial grid made by at least $1\times10^7 - 3\times10^7$(tens of millions) cells; time grid consisting of not less than $3\times10^4 - 5\times10^4$ time steps. A single simulation (i.e., one angle of incidence representative of 10 minutes in real scale) indicatively requires around 3 days on a 64 cores computational cluster;
- Models aiming to evaluate the static aerodynamic coefficients of a bridge deck without construction details (e.g., safety barriers), adopting a *non scale-resolving* turbulence model and a 2D computational domain in space: spatial grid composed at least by $1\times10^4 - 1\times10^5$ cells. If the flow is expected to be only slightly unsteady, it may be possible to perform time-independent RANS. On the contrary, if the flow is expected to be highly unsteady (e.g., due to vortex shedding), time-dependent URANS is mandatory with a time grid consisting of not less than $1\times10^4 - 1\times10^5$ time steps. The actual duration of the simulation shall be sufficient to allow the evaluation of the static aerodynamic coefficients removing the initialization phase. A single time-dependent simulation indicatively requires around 8 hours on a today two-cores computer.

These ranges, which are only indicative, can considerably vary depending on the geometry of the structure under investigation, on the details retained in the geometrical model, as well as on the turbulence modelling. Nevertheless, they clearly highlight the strong difference in the requirements which must be met in order to run simulations using different modelling approaches. The above indicative intervals can be suitably reduced for preliminary parametrical studies addressed to the aerodynamic optimization of the structure within the conceptual design stage.

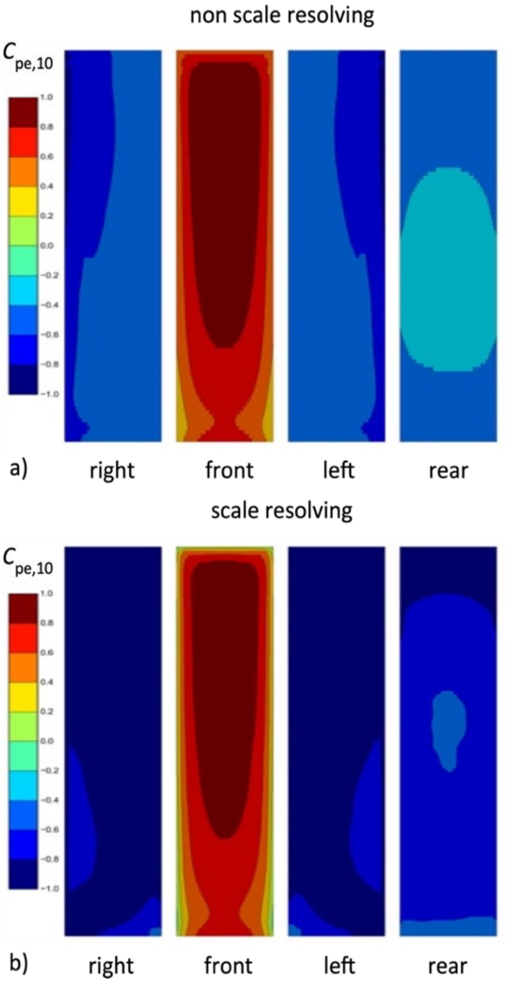


Fig. 16 Time-averaged C_p on tall building faces simulated by a *non scale-resolving* URANS *k*-*ω* (a) and a *scale-resolving* LES (b) turbulence model (after Fransos & Lo Giudice 2015)

5.3 Validating the model

As already clarified, validation is a fundamental activity in CWE and it represents an unavoidable prerequisite to proceed in any CWE study (side block in Fig. 9). In detail, the validation process:

• shall be consistent with well-established methodological indications for CFD validation, e.g., Roache *et al* (1986), AIAA (1998), ERCOFTAC (2000);

• shall verify that the specifically adopted computational model is able to reproduce the phenomena of interest and that it provides accurate results for the same *class of problem* the design case belongs to. In more explicit terms, the model validation is not strictly required for each single design case but needs to be performed as a preliminary activity for each *class of problem*, possibly considering more than one case belonging to the same *class of problem*;

• cannot be inferred in an indirect way by referring to technical documents or scientific studies by others, that refer to similar modelling components or numerical schemes. In other words, the validation shall be directly carried out by the *CWE Specialist* in charge of the study;

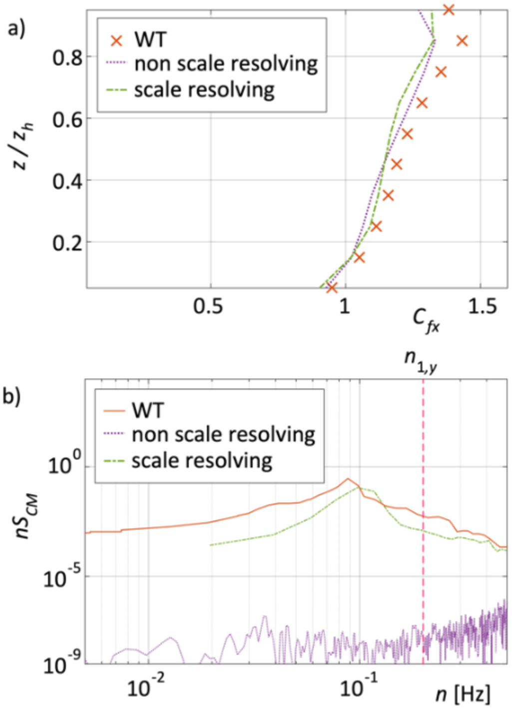


Fig. 17 Time-averaged drag coefficient at different relative height of a tall building (a), spectrum of the along-wind base moment (b) (URANS *k*-*ω* and LES after Fransos & Lo Giudice 2015)

• shall be carried out via a controlled, reproducible, and well-documented procedure.

Consequently, accounting also for the specific model components and for the *class of problems* under investigation, the validation phase shall be performed by comparing the computational results obtained using the model with available experimental results from WT tests, and/or with other high-fidelity computational simulations of proven validity.

Benchmark cases shall be selected by accounting for dominant factors that can contribute to the definition of the aerodynamic/aeroelastic structural behaviour, and aiming to evaluate the effectiveness of the computational model to reproduce their influence. It is recommended that:

• the adopted benchmarks are selected from scientifically and technically qualified sources. Databases of proven validity and completeness about the experimental setup and measurements are recommended, e.g., Ercoftac QNET-CFD Wiki, Tokyo Polytechnic University Aerodynamic Database, BARC benchmark database;

• the computational model includes and reproduces the same experimental setup in which the benchmarking measurements were obtained. The evidence of such an accurate reproduction shall be provided at the same time as the comparison of the results;

• the components of the modelling approach and/or the numerical approach might be varied, within the limits of the combinations characterized by proven validity, in order to evaluate their effect on the obtained results;

• the comparison among computational results and measurements shall not be limited to global quantities (e.g., global forces) and shall include local quantities (e.g., velocity and pressure fields), their statistical moments and other quantities of interest (e.g., mean, standard deviation, power spectral density). Comparing time-averaged integral quantities only (e.g., time-average values of the overall along-wind force), although of design interest, does not allow obtaining conclusive indications for validation purposes. As an example, Fig. 16 shows the comparison between the time-averaged pressure coefficient distribution over the tall building previously shown, obtained by means of *non scale-resolving* and *scale-resolving* models. A good overall agreement is observed and, as a consequence, Fig. 17(a) shows a good agreement between the two models also in terms of time-averaged integral drag force coefficient. However, Fig. 17(b) clearly shows that the spectra of the integral forces (e.g., the along-wind base moment in the example) are extremely different: the *non scale-resolving* model dramatically underestimates the energy content at any frequency, even though the simulation is time-dependent; conversely, the *scale-resolving* model provides results in reasonable agreement with WT experimental measurements.

5.4 Analyzing, reporting and independent checking of the CWE study

Once analyses are completed (third block in Fig. 9), the postprocessing of the results and the reporting of the study to the Customer are required (fourth block in Fig. 9). Such a task plays a major role because of its three-fold goal:

- providing results in a form immediately useful to the structural *Designer*. In such a perspective, the *CWE Specialist* shall identify *a priori* the results of interest for the analysis to be provided to the structural *Designer*, and plan in advance their storage during the computational runs. As an example, the tributary surfaces for each structural element shall be defined, and the computational domain may be partitioned accordingly in order to obtain for each of them the resultant wind force to be applied to the structure. As an example, in Fig. 12c each cladding panel of the facade is defined in order to provide the corresponding maximum peak pressure coefficient;
- providing a sound phenomenological reading of the results and the assessment of their robustness. The *CWE Specialist* shall pay particular attention to the physical interpretation of the results, and shall warn the *Customer*, if the *class of problem* includes or the simulation highlights:
  - curved surfaces, porous coatings, geometric features that are expected to potentially affect the overall wind flow but that cannot be reproduced with the desired level of detail at affordable computational costs;
  - an expected transition from laminar to turbulent flow;
  - a suspected coexistence of two or more different aerodynamic regimes close to each other in the space of the design parameters (bistable aerodynamic behavior of the structure).
- reporting all the study features that can allow not so much the complete reproducibility of the study by other *CWE Specialists*, as the independent check by the *Customer* him/herself, and/or possibly by a *Skilled Controller*, especially in the case of particularly challenging design cases. The *CWE Specialist* shall report and document all information necessary to assess the quality of the study:
  - main features of the class of problem;
  - all the components of the computational model, according to the *class of problem*;
  - type, name and version of the software and hardware used for the computational study;
  - validation study carried out for the computational setup used for the specific *class of problem*, justifying the adopted choices;
  - geometry of the computational domain, and specific data relevant to the computational grid (e.g., time step, resolution of the spatial grid at the solid walls and in other particularly significant part of the domain);
  - values of the reference quantities (e.g., lengths, areas, speeds) used for normalizing specific parameters (e.g., Reynolds number) and results (e.g., pressure coefficients, force coefficients, Strouhal number);
  - description of the simulation procedure;
  - characteristics of the incoming flow on the structure, in terms of spatial distributions of relevant quantities, also in relation to the adopted turbulence model;
  - verifications carried out to check the reliability of the simulation (e.g., accuracy of the pressure coefficients at stagnation point, undisturbed flow where actually expected);
  - evaluation of the convergence of the main statistics of some significant quantities for the specific *class of problem* with respect to the simulated time.

The details of the documentation summarizing these aspects shall be adequate for the purpose of the simulation and for the considered design phase. Once more, such postprocessing is analogous to the counterpart in WT experimental testing.

5.5 A tentative simplified mapping of CWE applications

As clearly emerges from the previous sections, the variability of the problems which can be studied using CWE, the numerous aspects which shall be considered in planning the simulations and the complexity of the computational model, make a schematic categorization of CWE studies extremely challenging, if not impossible. Indeed, such a categorization surely goes outside the purposes of the present paper, as well as the possibility to be included in *informative annexes*. In the authors' opinion it is nevertheless useful to provide a simplified map which includes some recurrent applications of CWE and frame them in the current state of the art.

To this purpose, with illustrative purposes only, we proceed by simplifying the *class of problem* definition from the five key aspects reported in Fig. 10 to only three. Namely we consider: (i) the structural type, (ii) the focus of the analysis and (iii) the design stage, which already allow providing a tentative framing of the possible use of CWE at the current state of the art.

As regard the description of the computational model, Fig. 2 proposes an articulation into seven main components.

Table 2 Reliability of the results obtained by CWE simulation according to the purpose of the analysis. SLS denotes Serviceability Limit States, ULS denotes Ultimate Limit States [H: High, M: Medium, L: Low, D: Discouraged, -: not envisaged].

PROBLEM CLASS	Approach to turbulence	Scale-resolving				Non scale-resolving			
	Design Stage	Preliminary		Detailed		Preliminary		Detailed	
	Limit State	SLS	ULS	SLS	ULS	SLS	ULS	SLS	ULS
Forces on Buildings		H	H	H	M	L	D	D	D
Forces on bridge decks		H	H	H	M	M	M	M	L
Local pressures		H	H	H	M	D	D	D	D
Vortex shedding		H	H	M	M	M	L	L	D
Galloping		H	H	M	L	M	L	L	L
Flutter		-	H	-	L	-	H	-	L
Comfort		H	-	H	-	M	-	M	-

Here, for illustrative purposes, we simplify the model specification taking into account a single aspect, namely the type of turbulence model i.e., *scale-resolving* or *non scale-resolving*. This is obviously a great simplification, but it already allows making broad subdivisions and, through the mutual compatibility between components, it actually sets numerous constraints on other components.

We hope that designers and, more generally, non *CWE Specialists* might find such map useful, despite the risk of premature aging and excessive simplification. In particular, Table 2 provides for the selected simplified *classes of problem* a qualitative judgment based on the authors' best knowledge regarding the application of CWE, as reported in CNR-DT R1-207/2018.

As it can be seen, the use of CWE is highly encouraged for preliminary studies, while more carefulness shall be obviously used for detailed ones. A fruitful use of *non scale-resolving* models can be envisaged, but their applicability is surely more limited than *scale-resolving* ones and strongly depends on the *class of problem*.

6. Conclusions

Exactly thirty years after both the Pozzati's words during his last lesson and the first CWE symposium, the paths of technical standards and Computational Wind Engineering are meeting in the field of structural design.

In this paper, the authors moved from the critical points risen by both an excess of prescriptive codification and an unconscious use of the rapidly evolving CWE techniques. Taking advantage of the support provided in drafting two recent recommendations and standards about the use of CWE in the structural design practice, the authors systematically addressed some crucial questions and proposed corresponding answers.

Does CWE need to face its wide and increasing use within the design practice through codification? Yes, the long-lasting total lack of codification opens the door to an unregulated CWE market driven by unscrupulous players.

Who shall be the main target of such a codification activity? At the current stage, in which *informative annexes* are provided, it appeared necessary to provide guidelines mainly addressing *Designers* rather than *CWE Specialists*. *What shall be its purpose?* The purpose shall be to ensure a timely and safe use of CWE in the design practice. A prescriptive approach to CWE codification appears to provide only limited protection against its misuse and it is condemned to premature obsolescence. For these reasons the authors preferred to use a performance-based approach.

Does codification increase the risk of a non-conscious and uncritical use of CWE by practitioners? Yes, if prescriptive codification is intended as a collection of black-box-like rules and protocols to be automatically applied by non *CWE Specialists* to carry out CWE simulations by themselves. No, if informative codification is intended to allow the structural *Designer* to dialogue and interact in an informed manner with the *CWE Specialist*. Overall, it is important to inform non *CWE Specialists* that CWE is not simply an algorithm implemented in software. Rather, CWE is a specialized multidisciplinary approach that results from joint competencies in Wind Engineering and Computational Fluid Dynamics, supported by long term professional experience. The concept of "virtual wind tunnel" as used in the commercial arena is completely misleading.

In a summary, considering that a single universal protocol to mix the components necessary to build an effective computational model does not exist, each specific design case shall be first ascribed to the *class of problem* it belongs to, according to its main features. Once the *class of problem* is identified, the multiple components of the CWE model shall be selected in order to secure proper interaction among them and overall efficiency. The resulting model shall be validated with respect to high fidelity data for each *class of problem*.

Acknowledgments

The activity described in this paper was carried out under the umbrella of the Italian Association for Wind Engineering (ANIV): the authors acknowledge its support.

Davide Fransos and Andrea Lo Giudice, former members of ANIV-CWE, contributed in the past to the group activities on CWE best practices and codification: the authors warmly thank them.

The Annex T of the Guide CNR-DT 207-R1/2018 was strongly encouraged by Giovanni Solari and Franco Maceri: the authors gratefully acknowledge their stimuli.

The Annex K of Eurocode 1–Part 1-4 has been first envisioned by Francesco Ricciardelli as member of the Project Team SC1.T3: EN 1991-1-4 (Wind) - Mandate M/515 Phase 2 tasks for the development of the 2nd generation of EN Eurocodes: the authors are grateful for his proposal to support the drafting of the part devoted to CWE.

The figures included in the paper partially refers to data, results and studies obtained by Others. Three nomenclatures are used in the figure captions: i. if data are gathered, analyzed and reworked by the Authors, hence the term "source" is used; ii. if results are taken form published scientific and technical literature, hence the term "after" accompanied by a bibliographic reference is used; iii. if results are taken by unpublished industrial studies, the term "Credits:" accompanied by the name of the Company owner of the rights is used.

It will be evident from the reference list that the authors are indebted to a large number of industrial and academic colleagues for much of the information that is included in this paper, and it is simply not possible to name them all. Their indirect valuable contribution is gratefully acknowledged.

References

Abu-Zidan, Y., Mendis, P. and Gunawardena, T. (2021) "Optimising the computational domain size in CFD simulations of tall buildings", *Heliyon*, **7**(4), e06723. https://doi.org/10.1016/j.heliyon.2021.e06723.

AIAA (1998) *Guide for the Verification and Validation of Computational Fluid Dynamics Simulations*, AIAA G-077–1998. https://doi.org/10.2514/4.472855.001.

AIJ-GWL (2005) "*AIJ Guide for Numerical Prediction of Wind Loads on Buildings*", Architectural Institute of Japan (AIJ), Tokyo, Japan.

ASCE/SEI 49-12 (1997) *Wind Tunnel Testing for Buildings and Other Structures*, American Society of Civil Engineers (ASCE).

ASCE 7-10 (2010) *Minimum Design Loads for Buildings and Other Structures*, American Society of Civil Engineers (ASCE).

ASCE 7-22 (2022) *Minimum Design Loads for Buildings and Other Structures*, American Society of Civil Engineers (ASCE).

ASCE (1985), *Aerodynamics Committee*, *Proposed Manual of Practice for Wind Tunnel Testing of Buildings and Structures*, ASCE Spring Convention, Denver, Colorado.

Ballio, G. and Solari, G. (1988) "The new Italian recommendation for wind loads on structures: Basic assumptions and critical considerations", *J. Wind Eng. Ind. Aerod*, **30**(1-3), 123-132. https://doi.org/10.1016/0167-6105(88)90077-3.

BARC benchmark database, www.aniv-iawe.org/barc-data/

Bearman, P.W. and Morel, T. (1983), "Effect of free stream turbulence on the flow around bluff bodies", *Prog. Aerosp. Sci.*, **20**(2-3), 97-123. https://doi.org/10.1016/0376-0421(83)90002-7.

Belostotsky, A.M., Akimov, P.A. and Afanasyeva, I.N. (2019), "About 'Legitimization' of numerical modelling of wind impacts on buildings and structures", *Int. J. Comp. Civil Struct. Eng.*, **15**(4), 14-24. https://doi.org/10.22337/2587-9618-2019-15-4-14-24.

Bertani, G. Patruno, L. and Gandia, F.A (2022), "Low-fidelity simulations in Computational Wind Engineering: shortcomings of 2D RANS in fully separated flows", *Wind Struct.*, **34,** 499-510. https://doi.org/10.12989/was.2022.34.6.499.

Blocken, B. (2014), "50 years of Computational Wind Engineering: Past, present and future", *J. Wind Eng. Ind. Aerod.*, **129**, 69-102. https://doi.org/10.1016/j.jweia.2014.03.008.

Bruno, L., Fransos, D., Coste, N. and Bosco, A. (2010), "3D flow around a rectangular cylinder: A computational study", *J. Wind Eng. Ind. Aerod.*, **98**, 263-276. https://doi.org/10.1016/j.jweia.2009.10.005.

Bruno, L. and Oberto, D. (2022), "Effects of cell quality in grid boundary layer on the simulated flow around a square cylinder", *Comp. Fluids*, **238**, 10535. https://doi.org/10.1016/j.compfluid.2022.105351.

Bruno, L., Salvetti, M.V. and Ricciardelli, F. (2014), "Benchmark on the Aerodynamics of a Rectangular 5:1 Cylinder: An overview after the first four years of activity", *J. Wind Eng. Ind. Aerod.*, **126**, 87-106. https://doi.org/10.1016/j.jweia.2014.01.005.

Cammelli, S., Dorigatti, F., Hackett, D., Ping To, A. and Vazquez, B. (2022), *A Position Paper on Experimental and Computational Methods in Wind Engineering*, UK Wind Engineering Society (WES), London.

Canuto, C., Hussaini, M. Y., Quarteroni, A. and Zang, T. A. (1988), *Spectral Methods in Fluid Dynamics*. Part of the book series: Scientific Computation (SCIENTCOMP), Springer-Verlag Berlin Heidelberg. https://doi.org/10.1007/978-3-642-84108-8.

Castro, I.P. and Graham, J.M.R. (1999), "Numerical wind engineering: the way ahead?", *Proc. Inst. Civil Eng. – Struct. Build.*, **134**(3), 275-277. 10.1680/istbu.1999.31569.

Cermak, J.E. (1975), "Application of Fluid Mechanics to Wind Engineering", Freeman Scholar Lecture, *J. Fluid Eng. ASME*, **97** (1). https://doi.org/10.1115/1.3447225.

Choi, C.K. and Kwon, D.W. (1998), "Wind tunnel blockage effects on aerodynamic behavior of bluff body", *Wind Struct.*, **1**(4), 351-364. https://doi.org/10.12989/was.1998.1.4.351.

CNR-DT 207/2008 (2008) *Guide for the Assessment of Wind Actions and Effects on Structures*, National Research Council of Italy - Advisory Committee on Technical Recommendations for Construction; Rome, Italy.

CNR-DT R1-207/2018 (2018) *Guide for the Assessment of Wind Actions and Effects on Structures*, National Research Council of Italy - Advisory Committee on Technical Recommendations for Construction; Rome, Italy.

Dorney, S.M. (2003), "CFD Process pre- and post-processing automation in support of space propulsion", *39th AIAA/ASME/SAE/ASEE Joint Propulsion Conference and Exhibit*, Huntsville, AL, 20-23 July 2003.

EN 1991-1-4 (2005), *Eurocode 1: Actions on Structures*, *Part 1-4: General Actions*, *Wind Actions*, European Committee for Standardization (CEN); Brussels, Belgium.

ERCOFTAC (2000), *Best Practices Guidelines for Industrial Computational Fluid Dynamics*, Version 1.0, January 2000.

Ercoftac QNET-CFD Wiki, http://qnet-ercoftac.cfms.org.uk/w/index.php/Main_Page.

Ferziger, J.H. (1993), "Estimation and reduction of numerical error" FED vol. 158, In: *Proceedings of the Symposium on Quantification of Uncertainty in Computational Fluid Dynamics*, ASME Fluid Engineering Division, Summer Meeting, Washington DC, June, 1-8.

Fransos, D. and Lo Giudice, A. (2015), "On the use of computational simulation in the determination of wind loads on structures: design experiences and food for thought", *Proceedings of the XXV Congress CTA*, Salerno, October.

Hirsch, C. (2007), *Numerical Computation of Internal and External Flows: The Fundamentals of Computational Fluid Dynamics*. Elsevier. https://doi.org/10.1016/B978-0-7506-6594-

0.X5037-1.

Holmes, J.D. and Carpenter, P. (1990), "The effect of Jensen number variations on the wind loads on a low-rise building", *J. Wind Eng. Ind. Aerod.*, **36**, 1279-1288. https://doi.org/10.1016/0167-6105(90)90124-U.

Houghtno, E.L. and Carruthers, N.B. (1976), *Wind Forces on Buildings and Structures: An Introduction*. John Wiley & Sons Inc.

ISO 4354:2009 (2009), *Wind Actions on Structures*. *ISO/TC 98/SC 3 Loads*, *Forces and Other Actions*. International Organization for Standardization, Geneva, Switzerland.

Italian National Research Council (1985), *CNR10012/85: Guide for the Assessment of Wind Actions on Buildings*, Rome, 1985.

Lamberti, G. and Gorlé, C. (2020). "Sensitivity of LES predictions of wind loading on a high-rise building to the inflow boundary condition", *J. Wind Eng. Ind. Aerod.*, **206**, 104370. https://doi.org/10.1016/j.jweia.2020.104370.

Leschziner, M.A. (1990), "Modelling engineering flows with Reynolds stress turbulence closure", *J. Wind Eng. Ind. Aerod.*, **35**, 21-47. 10.1016/0167-6105(90)90209-U.

Liu, H. (1990) *Wind Engineering: A Handbook for Structural Engineering*. Pearson Education.

Menter, F.R. and Egorov, Y. (2010), "The scale-adaptive simulation method for unsteady turbulent flow predictions. Part 1: theory and model description", *Flow*, *Turbulence Combustion*, **85**, 113-138. 10.1007/s10494-010-9264-5.

Mod2-SP20.13330.2016 (2019) *Izmenenie No. 2 k SP 20.13330.2016 "SNIP 2.01.07-85* Nagruzki i vozdejstvija"* [The Second Modification to SP 20.13330.2016 "SNIP 2.01.07-85* Loads and impacts"], Ministry of Construction and Housing and Communal Services of the Russian Federation, Moscow, Russian Federation.

Moukalled, F., Mangani, L. and Darwish, M. (2016), *The Finite Volume Method in Computational Fluid Dynamics*. Springer, Cham. 10.1007/978-3-319-16874-6.

Murakami, S. (1990), "Computational wind engineering", *J. Wind Eng. Ind. Aerod.*, **36**, 517-538. https://doi.org/10.1016/0167-6105(90)90335-A.

Murakami, S. (1997), "Current status and future trends in computational wind engineering", *J. Wind Eng. Ind. Aerod.*, **67-68**, 3-34. 10.1016/S0167-6105(97)00230-4.

Murakami, S. (1998), "Overview of turbulence models applied in CWE-1997", *J. Wind Eng. Ind. Aerod.*, **74-76**, 1-24. https://doi.org/10.1016/S0167-6105(98)00004-X.

Murakami, S. (1993), "Computational Wind Engineering 1", *Proceedings of the 1st International Symposium on Computational Wind Engineering (CWE 92)*, Tokyo, Japan, August. https://doi.org/10.1016/C2009-0-10273-8.

Noda, H. and Nakayama, A. (2003), "Free-stream turbulence effects on the instantaneous pressure and forces on cylinders of rectangular cross section", *Experiments Fluids*, **34**(3), 332-344. https://doi.org/10.1007/s00348-002-0562-0.

NF EN 1991-1-4/NA:2008-03 (2008) *French National Annex to EN 1991-1-4:2005 — General actions — Wind actions*. Association Française de Normalisation (AFNOR), La Plaine Saint-Denis, France.

NTC-2018 (2018) *Italian Building Code*, Italian Ministry of Infrastructure, Rome, Italy.

Oberkampf, W.L., Trucano, T.G. and Hirsch, C. (2004), "Verification, validation, and predictive capability in computational engineering and physics", *Appl. Mech. Rev.*, **57**(5), 345-384. https://doi.org/10.1115/1.1767847.

Oberkampf, W.L. and Trucano, T.G. (2002), *Verification and Validation in Computational Fluid Dynamics*, Sandia National Laboratories Report, SAND2002-0529.

Patruno, L. and de Miranda, S. (2020). "Unsteady inflow conditions: A variationally based solution to the insurgence of pressure fluctuations", *Comput. Methods Appl. Mech. Eng.*, **363**, 112894. https://doi.org/10.1016/j.cma.2020.112894.

Paya-Zaforteza, I. and Garlock, M.E.M. (2021), "Structural Engineering Heroes and Their Inspirational Journey", *Struct. Eng. Int.*, **31**(4) 584-597. https://doi.org/10.1080/10168664.2021.1919038.

Peyret, R. and Taylor, T.D. (1983), *Computational Methods for Fluid Flow* in: Scientific Computation (SCIENTCOMP), Springer. https://doi.org/10.1007/978-3-642-85952-6.

Pope, S.B. (2000), *Turbulent Flows*. Cambridge university press. 10.1017/CBO9780511840531

Potsis, T., Tominaga, Y. and Stathopoulos, T. (2023), "Computational wind engineering: 30 years of research progress in building structures and environment", *J. Wind Eng. Ind. Aerod.*, **234**, 105346. https://doi.org/10.1016/j.jweia.2023.105346.

Pozzati, P. (1992) *Proliferation of Codes and Technicism*, last official lecture of the Course on Structural Design, academic year 1991-1992, Faculty of Engineering, University of Bologna, June 3rd 1992, Bologna (in Italian).

PrEN 1991-1-4:2021 (2021), *Eurocode 1: Actions on Structures - Part 1-4: General Actions - Wind Actions*, European Committee for Standardization (CEN); Brussels, Belgium.

Regalado, A. (2011), Who Coined 'Cloud Computing'?, MIT Technology Review, https://www.technologyreview.com/2011/10/31/257406/who-coined-cloud-computing/

Ricciardelli, F. (2023), "prEN1991-1-4:2021: the draft Second Generation Eurocode on wind actions on structures - a personal view", *Wind Struct.*, **37**(2)

Richards, P.J. and Norris, S.E. (2019) "Appropriate boundary conditions for computational wind engineering: Still an issue after 25 years", *J. Wind Eng. Ind. Aerod.*, **190**, 245-255. https://doi.org/10.1016/j.jweia.2019.05.012.

Roache, P.J. (1997), "Quantification of uncertainty in computational fluid dynamics", *Annu. Rev. Fluid Mech.*, **29**, 123-160. https://doi.org/10.1146/annurev.fluid.29.1.123.

Roache, P.J., Ghia, K., White, F. (1986) "Editorial policy statement on the control of numerical accuracy". ASME. *J. Fluids Eng.*, **108** (1), 2. https://doi.org/10.1115/1.3242537.

Simiu, E. and Scanlan, R.H. (1996), *Wind Effects on Structures*, Dover Pubblications. 10.1002/9781119375890.

Solari, G. (1993), "Gust buffeting. I: Peak wind velocity and equivalent pressure", *J. Struct. Eng.*, **119**, 365-382. https://doi.org/10.1061/(ASCE)0733-9445(1993)119:2(365).

Spalart, P.R. (2001), *Young–Person's Guide to Detached-Eddy Simulation Grids*, NASA CR-2001-211032.

Stathopoulos, T. (2002), "The numerical wind tunnel for industrial aerodynamics: real or virtual in the new millennium?", *Wind Struct.*, **5**(2-4), 193-208. https://doi.org/10.12989/was.2002.5.2_3_4.193.

Tamura, T., Nozawa, K. and Kondo, K. (2008), "AIJ guide for numerical prediction of wind loads on buildings", *J. Wind Eng. Ind. Aerod.*, **96**(10), 1974-1984. https://doi.org/10.1016/j.jweia.2008.02.020.

Tamura, Y. and Van Phuc, P. (2015), "Development of CFD and applications: Monologue by a non-CFD-expert", *J. Wind Eng. Ind. Aerod.*, **144**, 3-13. https://doi.org/10.1016/j.jweia.2015.05.003.

Thordal, M.S., Bennetsen, J.C. and Koss, H.H.H. (2019), "Review for practical application of CFD for the determination of wind load on high-rise buildings", *J. Wind Eng. Ind. Aerod.*, **186**, 155-168. https://doi.org/10.1016/j.jweia.2018.12.019.

Tokyo Polytechnic University Aerodynamic Database, http://wind.arch.t-kougei.ac.jp/system/eng/contents/code/tpu

Tominaga, Y. and Stathopoulos, T. (2013), "CFD simulation of near-field pollutant dispersion in the urban environment: a review of current modelling techniques", *Atmos. Environ.*, **79**, 716-730. https://doi.org/10.1016/j.atmosenv.2013.07.028.

Thompson M.K. and Thompson, J.M. (2017), *ANSYS Mechanical APDL for Finite Element Analysis*, Elsevier

Trias, F.X., Gorobets, A. and Oliva, A. (2015), "Turbulent flow around a square cylinder at Reynolds number 22000: A DNS study", *Comput. Fluids*, **123**, 87-98. https://doi.org/10.1016/j.compfluid.2015.09.013.

Versteeg, H.K. and Malalasekera, W. (2007), *An Introduction to Computational Fluid Dynamics: The Finite Volume Method*. Pearson education.

Wilcox, D.C. (1998), *Turbulence Modeling for CFD*. La Canada, CA: DCW Industries.

Wu, X. (2017), "Inflow turbulence generation methods", *Annual. Rev. Fluid Mech.*, **49**, 23-49. https://doi.org/10.1146/annurev-fluid-010816-060322.

Xing, J., Patruno, L., Pozzuoli, C., Pedro, G., de Miranda, S. and Ubertini, F. (2022), "Wind loads prediction using LES: Inflow generation, accuracy and cost assessment for the case of Torre Gioia 22", *Eng. Struct.*, **262**, 114292. https://doi.org/10.1016/j.engstruct.2022.114292.