Systemic Analytics

www.sa-journal.org

Syst. Anal. Vol. 3, No. 1 (2025) 35–58.

Paper Type: Original Article

Aerodynamic Optimisation of Tall Buildings in Extreme Wind Condition with CFD Using Ansys Fluents: Evidence from Literature

Akeem Babatunde Animasaun[1,*], Matthew Oluwatosin Adewuyi[1], Malik Adesokan Sanusi[1]

[1] Department of Civil and Structural Engineering, Teeside University, UK; animasaunakeem@gmail.com.

Citation:

Received: 17 November 2024 *Revised: 21 January 2025* *Accepted: 11 March 2025*	Animasaun, A. B., Adewuyi, M. O., & Sanusi, M. A. (2025). Aerodynamic optimisation of tall buildings in extreme wind condition with CFD using ansys fluents: Evidence from literature. *Systemic Analytics*, *3*(1), 35-58.

Abstract

There has been a noticeable increase in the adoption of Computational Fluid Dynamics (CFD) for wind design on tall structures within the last 20 years. Many tudies have been carried out on CFD with validation at the forefront of the majority of them. The tall building designs have not adequately addressed the difficulties posed by CFD and the various analytical techniques. Additionally, there are not many essential best practice standards for employing CFD for wind study of buildings that researchers and practitioners may easily follow. A thorough technical examination of the adoption of CFD on tall buildings and structures is presented in this research to achieve this goal. Detailed examination of CFD and its related design issues that are unique to tall building design, include domain and mesh configurations, inflow turbulence, and turbulence model. Additionally, a thorough literature review of CFD research on tall structures is provided, encompassing all significant subjects from intricate Fluid-Structure Interaction (FSI) studies to simple bluff body aerodynamics. In this study, appropriate literature are critically assessed (objectively) and contrasted with arguments in favour of it drawn from the perspective of author. This study concludes by discussing possible future numerical techniques such as the Lattice-Boltzmann Method (LBM) and the adoption of Artificial Intelligence (AI) to tall skyscraper design.

Keywords: Computational fluid dynamics, Fluid-structure interaction, Tall buildings, Wind loading building design.

1 | Introduction

1.1 | Background

The height and slenderness have made modern tall structures prone to wind-induced vibrations. Tall structures are now made of sturdy, lightweight materials as a result of recent developments in material science

Corresponding Author: animasaunakeem@gmail.com

10.31181/sa31202542

and construction, which increase the effects of wind-induced vibrations [1]. Due to the fact that slender tall buildings endure low frequency and low acceleration building vibrations that impact occupant comfort, structural serviceability is one of the primary challenges [2]. Additionally, the lateral load that governs the design of structural integrity is wind [2].

The industry standard for assessing wind-sensitive structures, such tall buildings, is wind tunnel testing. Over the past century, wind tunnel testing has advanced as methods for carrying out experiments in a scaled-up setting have been proposed and continuously assessed to enhance the methodology. Over the years, many researchers have significantly advanced wind tunnel techniques. The significance of scaling rules and the necessity of doing experiments under boundary layer circumstances were first mentioned by [3] Since then, wind tunnel testing has changed in terms of testing procedures, equipment, and acquisition strategies; Leighton and Cochran have thoroughly described and recorded these changes [4].

Three main methods are used in wind tunnel practice today to evaluate wind-sensitive buildings:

I. The High-Frequency Base Balance (HFBB) method [5].

II. The High-Frequency Pressure Integration (HFPI) method [6].

III. The more complex and conventional aeroelastic modeling method which in certain situations can also measure surface pressures [7].

The most often used wind tunnel techniques are the HFBB and HFPI methods, particularly for measuring aerodynamic forces and surface pressure caused by wind, respectively. Assuming a linear building mode form, the HFBB technique employs a spectral approach to calculate the generalized forces acting on the building [8]. The capacity to precisely record forces and moments at the base is the primary benefit of the HFBB approach. It is unable to identify cladding stresses, though. The HFBB approach has evolved since its initial use, allowing for the inclusion of higher-order mode influence [9] and mode shape modifications [10] Despite these developments, the HFBB approach still presents challenges throughout the model production phase.

It is challenging for thin tall models to reproduce the low weight and stiffness required for HFBB models. The HFBB setup's resonant effects result from the inability to generate such a model, and this might taint the data of the forces obtained at the base [8]. The test arrangement is limited to "lower wind speeds where the Reynolds number difference" [11] might impact the "overall results since high wind speeds can further induce resonance effects" [6]. Additionally, it is frequently discovered that the effect of higher modes including linked modes of vibration is crucial for tall buildings and cannot be disregarded. As a result, the HFBB technique's limitations make it unsuitable for application in the construction of tall, thin buildings [6].

In contrast, the HFPI technique can detect vertical profiles and cladding pressures, but it cannot measure base moments and forces as well as the HFBB (precision rises with resolution of pressure taps). The HFPI approach may accurately anticipate structural reaction by predicting responses for all modes of vibration, provided that there is an adequate distribution of pressure taps [12]. The HFPI model is easy to build in terms of requirements because it must be stiff and has no mass (or model flexibility) limits. The capacity to measure higher resolution surface pressure maps is directly related to the accuracy of the HFPI approach, and this is frequently constrained by apparatus and surface complexities, particularly of tall, thin structures.

Recent research has successfully used the HFPI approach to determine the structure's response using modal analysis [13]. Multi-Degree of Freedom (MDOF) aeroelastic models have been more popular in recent years for designing tall, thin structures [14]. Since it can measure all known structural reactions, the aeroelastic approach is the most thorough experimental technique used in wind tunnel design today [15], [16]. All vibrational modes should be able to be replicated by an MDOF aeroelastic model with sizable modal participation factors that are essential to the structure's response.

The "aeroelastic model" is difficult and expensive to manufacture given the requirements and precise representation of the structure, particularly in situations where complex vibration modes need to be taken into account. As a result, the "aeroelastic model" is typically used in the later stages of design and is frequently saved for complicated structures [2]. An alternative to an "MDOF aeroelastic model" is the stick aeroelastic model. Time and money can be saved since it is a Single Degree of Freedom (SDOF) model that just replicates the two basic sway patterns of vibration [17]. Only structures with dominating basic sway modes that is, moderately tall buildings with regular geometry can be used with the stick aeroelastic model. Even so, it has been discovered to produce wildly.

1.2|Potentials of Numerical Modelling Using CFD

The need for MDOF aeroelastic models has increased as a result of growing research interest in tall, thin structures. When higher-order modes become important and must be incorporated, the complex process of creating aeroelastic models frequently results in manufacturing challenges at the model size. This may be a constraint when thinking about experimental models, particularly because structures are typically more intricate. Researchers and designers have looked at the prospect of employing numerical modeling as an alternative to forecast wind-induced stresses on tall buildings, given the recent developments in computer hardware and technology. Leading the way in numerical modeling is Computational Fluid Dynamics (CFD), which frequently uses the Finite Volume Method (FVM) for discretization in the majority of commercial programs such FLUENT [18], [19].

Computational Wind Engineering (CWE) is the adoption of numerical and computational techniques to wind engineering issues [20], and CFD has been a key player in this area. Only a small part of a larger endeavor in CWE that addresses a wide range of issues is the study of CFD on tall and tall structures. Because of its adaptability and computational capacity to forecast flow phenomena, CFD is becoming more and more popular in the field of CWE. Due to recent advancements in the research community, CWE has effectively moved from an emerging to an established subject in wind engineering [20].

In urban settings, topics including pollution dispersion [21-23], building comfort analysis [23], [24], and Pedestrian Level Wind (PLW) [25-27] are already well-established and have demonstrated that using CFD can yield high numerical accuracy. The computational effort needed has significantly decreased recently due to advancements in turbulence modeling and computer technology. The capacity to conduct many parametric studies and quantify data inside a spatial volume without being restricted to instrumentation like that found in an experimental context is arguably the greatest benefit of such an approach. External aerodynamics and the flow fields surrounding the bluff bodies were examined in some of the first structural applications of CFD on regular forms [23], [25].

"Validations in terms of stagnation points, pressure quantities, and velocity fields (or vectors) and their distribution in their close and distant areas were measured in these early experiments, which were conducted for 2D cross-sections" [26-28]. At the forefront of these early CFD implementations were validation studies, which focused on the suitability of various turbulence models for simulating flow processes and compared them with experimental results. Reynolds Averaging Navier Stokes (RANS) models were used instead of scale-resolving turbulence models like Large Eddy Simulations (LES), which were judged to be too costly [27], [28]. Due to hardware limitations, a variety of RANS turbulent models were developed, however they were only able to offer statistical numbers and were unable to resolve stochastic turbulent movements of flow [29], [30].

In their early writings, Rodi [31] and Murakami [32] provide a clear description of these early investigations and their findings with regard to open flows for bluff bodies. At the time, Rodi [31] came to the significant conclusion that RANS turbulence models will be essential in engineering applications because of their practicality, regardless of their correctness. Additionally, the significance of LES simulations and its capacity to provide precise simulations in terms of both statistical and transient flow properties were covered by Rodi [31] and Murakami [32]. They did, however, come to the conclusion at the time that, even while LES simulations are correct, they are computationally costly for big flow issues with high Reynolds number flows. According to Rodi [31], LES will be widely employed in the "foreseeable but more distant future" as computer hardware advances.

The "estimation of wind loads and flow phenomena for bluff bodies has been the subject of a wide range of study over the past forty years" [16], building on the work of Rodi [31] and Murakami [32]. Examples of studies that have examined the development of CFD techniques on bluff body aerodynamics, such as square and circular cylinders, are those by Lübcke et al. [33], Rodi [34], Kravchenko [35], and Catalano et al. [33]. More significantly, it was noted that different turbulence modeling methodologies were expanding quickly. One prominent example is the k − omega shear stress transfer (SSTkω) [31], [32], [36-38] RANS turbulence model, which was developed to precisely forecast wall-bounded flows - a shortcoming seen in k - ε type models.

Additionally, hybrid RANS-LES models were used to describe scale-resolving flow features at a reduced computing cost, such as Delayed-DES (DDES) and the Detached Eddy Simulation (DES) [39-41]. The CFD numerical studies on the aerodynamics of bluff bodies were crucial since the majority of the results were applied to civil construction structures [32], [35], [42]. Low-rise buildings with straightforward geometry, such the Texas Tech University (TTU) Building, were the primary focus of some of the earliest CFD applications on buildings [43]. Naturally, the results of these early CFD adaptations for low-rise structures carried over to tall structure modeling [44], [45].

Huang et al. [46] conducted the first and arguably most significant investigation using CFD on tall structures in 2007. The standard CAARC building was used for the investigation, and an LES model with synthetic turbulence and Unsteady-RANS (URANS) models were examined [47]. The study found problems with RANS models' shortcomings, particularly when it comes to large structures and the measurement of surface pressure and flow streamlines. More importantly, it was shown that mean and standard deviation pressure were influenced by velocity and turbulence intensity patterns. Additionally, they showed that LES simulations might yield accurate estimates of wind-induced quantities, although they came to the conclusion that further study was required.

Nearly 20 years of CFD research on tall structures were made possible by Huang et al.'s [46] work. Since then, academics have looked at a variety of tall building design-related problems, most of which were carried out as parametric validation studies. These themes include the impacts of shielding, corner adjustments, and building shape modifications. More significantly, studies of the suitability of different turbulence models, grid generation strategies, and inflow turbulence approaches have been conducted over time with an emphasis on the construction of tall buildings.

Important features of tall building design, including experimental and numerical techniques, are covered in recent literature reviews on tall structures by Hou and Jafari [48], Jafari and Alice [38], Abu-zidan et al. [49], and Thordal et al. [50]. None of these publications, however, goes into great detail about how CFD has developed or the modeling difficulties it has when dealing with big structures. In order to highlight the advancements and noteworthy successes in the application of CFD to tall structures, as well as the potential for the future, this book attempts to summarize the research efforts conducted over the last many decades. The difficulties that contemporary modeling approaches confront will also be covered, including mesh generation methods, Reynolds number impacts, the significance of turbulence modeling, the creation of inflow profiles, sub-grid scale modeling, etc.

More significantly, it is also addressed how the most advanced Fluid-Structure Interaction (FSI) techniques may be used to directly assess structural reaction. The findings of various researchers shall, if feasible, be critically assessed and accompanied with arguments in favor of them. Lastly, based on the authors' vast expertise with CFD and tall structure design, every attempt has been made to present an impartial and thorough overview of all works pertaining to CFD applications on tall buildings.

2 | Methods

This study relied on detailed literature review form of methodology regarding the examination of the adoption of CFD on tall buildings and structures. The literatures are critically assessed (objectively) and contrasted with arguments as drawn from the author's experience.

2.1|Numerical Modelling and Its Challenges

The crucial numerical modeling factors that must be taken into account when using CFD on tall buildings are discussed in this section. There is discussion of domain and mesh design, inflow turbulence production, and turbulence model selection. This section explores the numerical modeling issues and difficulties related to these four crucial factors, which are thought to be at the core of CFD numerical modeling for tall structures. As a result, this part provides comprehensive literature on the development of these important components along with some suggestions. The ultimate goal is to inform the reader of the fundamental factors to be taken into account before selecting a suitable model for the given issue.

2.2|Turbulence Modeling

The "Navier-Stokes (N-S) equations are used in CFD to solve the finite element (FE) code for a given flow problem inside a spatially discretized volume". The "degree of complexity of the simulated flow is determined by how well the turbulence model solves the N-S equations". Turbulence models are used to explain fluid flow inside an enclosed region. Directly solving the N-S equations is not practical for the majority of engineering applications that take into account high Reynolds (Re) number flows because it requires a significant amount of computing power to resolve the whole spectrum of turbulence length scales. This type of simulation is sometimes called Direct Numerical Simulation (DNS), and it typically requires a CPU time of Re3 and scales in number of grid points as a function of Re9/4 [51], [52].

For comparison, a DNS simulation that captures all turbulence scales would need about 180 billion grid cells, according to Huang et al. [46] assessment of the CAARC building (where Re > 105). Even at the model size, "it is clear that DNS cannot be implemented in the context of tall building design" . Since "turbulence is frequently modeled, the complexity of the turbulence model used determines how well turbulent scales are resolved" [50]. Because of this, "a hierarchy of turbulence models may be arranged, with 1st order RANS turbulence models (which mimic all flow scales) at the bottom and DNS and LES at the very top in terms of complexity" [50], [51].

It is important to acknowledge that DNS applications are becoming more and more common in recent research [45]. A cube submerged in laminar and turbulent boundary layers, for example, had a maximum Reynolds (Re) number of 3000 (based on cube height and freestream velocity) in 3D-DNS simulations performed by Diaz-Daniel et al. [53]. They examined downstream-generated flow configurations and contrasted them with experimental results. Chen and colleagues [54] used 3D-DNS to investigate the effects of a circular cylinder that was positioned close to a stationary wall at a subcritical Reynolds (Re) number of 500. Chen et al. [55] examined the "SDOF FSI interference effects of two circular cylinders with different gap ratios in a different investigation". They described "the flow characteristics of the observed cylinder oscillations using 3D-DNS simulations" [11].

Lastly, Zhao et al. [56] used "3D-DNS simulations at a maximum Reynolds (Re) number of 1000 to investigate interference effects between two rectangular cylinders at varying width ratios". The primary goal of all these experiments was to examine basic flow mechanics, where 3D-DNS can provide more clarity on significant scale aspects. It should be highlighted, nonetheless, that practically all of the 3D-DNS simulations mentioned are examined in smaller domains and at lower Reynolds (Re) aate large-scale eddie formations, Tominaga [57] conducted URANS simulations on a tall structure (rectangular cylinder) using the SSTkω, Skε, RNG kε, and Realisable kε (Rkε) turbulence models. It was demonstrated that the unstable fluctuation behind the structure could be replicated by the SSTkω. Nevertheless, the SSTkω models overestimated separation at the building corner because they underestimated Turbulent Kinetic Energy (TKE).

In order to replicate adequate TKE that is equivalent to experimental results, Tominaga [57] suggested changing the Epsilon term in the RNG kε. The reattachment lengths on the roof and behind the building were still overstated by the URANS simulation in spite of these modifications. In research by Mohotti et al. [64], Steady-RANS (SRANS) turbulence models were applied to a tall structure with circular cross-sections and an unconventional shape. As opposed to separation at leading edges, as is seen in square or rectangular structures, the geometry's circularity meant that flow separation happened as a result of an unfavourable pressure gradient. The turbulence model and the specific flow problem determine the value of y+ in most cases.

LES resolves all scales of turbulent flows to the mesh size from the standpoint of numerical modelling [65]. To give enough resolution, the mesh size must be carefully chosen, particularly in areas surrounding the building surface. Most people agree that the initial cell height at the boundary should have enough inflation layers and a suitable growth rate when using LES for tall buildings. As a result, the SGS model is crucial, particularly regarding the boundary and viscous eddies. The Smagorinsky-Lilly SGS model [58] (S - model), which assumes a single model constant (Cs) to simulate the eddy viscosity, is one of the fundamental SGS models frequently seen in commercial codes.

Germano et al. [59] and Lilly [60] extended this model, pointing out that Cs are dynamically determined (D - model) and that a single universal constant could not be employed for LES modelling. The dynamic kinetic energy model (Dksgs - model), which was put out by Kim and Menon [61], is an additional modification of the D-model that enhances it by considering TKE. The S-model was used in the first LES implementations on bluff bodies by Rodi [31], [34]. However, Murakami [62] demonstrated that eddy viscosity stresses could not be effectively evaluated using a unique constant.

To adequately recreate different flow properties, different Cs values were needed [62]. Different model constants were required for each scenario, including downstream free shear flow, recirculation of flow, and approaching flow [52], [62]. It was inappropriate to resolve SGS stresses with a single constant since towering structures are prone to intricate flow phenomena. In recent literature, dynamic models for tall structures have been applied to the CAARC building [64], [65] and square/rectangular prisms [62], [64], [65]. However, the Wall Adapting Local Eddy (WALE) SGS model was found to be most appropriate in a thorough analysis conducted by Wijesooriya et al. [66], which compares five SGS models.

It was demonstrated that the WALE model could most accurately forecast the point of flow separation for a tall structure with circular cross-sections. Tall structures' dynamic sensitivity and the stochastic nature of wind necessitate LES-type modelling that can replicate transient wind behaviour. However, the same cannot be produced using RANS turbulence models. The Power Spectral Density (PSD) curves of crosswind forces for LES and MMK kɛ were compared by Huang et al. [46] with experimental results from Obasaju [67]. The RANS MMK kɛ can only predict the crosswind force's peak frequency based on the PSD curve; Tominaga [57] reported a similar finding. By contrast, the LES model is more accurate in reproducing the PSD curve across all frequencies. An essential modelling need in CFD simulations for tall structures, the inflow turbulence generator, maybe the cause of the differences between the PSD curve for LES and experimental.

3 | Inflow Turbulence Generation

The "numerical simulation of atmospheric boundary layer conditions is one of the major problems with CFD modeling" [11]. Similar to real wind tunnel testing, "the numerical domain must be able to replicate the power spectra, spatial coherence, wind profile, turbulence intensity profile, and spatial correlation statistics" [4]. Thordal et al. [50] have specified many methods for producing inflow turbulence, including

I. Wind tunnel replication,

II. Domain recycling.

III. Precursor domain method.

IV. Synthetic turbulence.

For a few reasons, the "later synthetic turbulence inflow method is the most popular of these four approaches" [68]. The "size of the domain and the complexity of the flow - which is produced by the impediments positioned (sometimes in the shape of blocks) to create turbulence-make methods (a), (b), and

(c) more computationally demanding" [69]. The "mesh density used at the barriers has a significant impact on the resulting turbulence. To adequately resolve the "scales down to the inertial subrange, a strict mesh density can be required because the blocks are typically smaller than the goal construction" [52]. Additionally, the complexity and character of the flow may make it impossible for the simulation to reach convergence within a time step, necessitating the adoption of a more reliable pressure-velocity coupling technique.

Thordal et al. [70] provide an example of precursor domain application and wind tunnel domain replication, seeing domain mesh sizes exceeding 30 million elements. Inflow turbulence is produced via synthetic turbulence techniques, which define an algorithm capable of producing random fluctuations. The "Fourier Transform-based method, like the Random Flow Generation (RFG)" of Smirnov et al. [71], which is an adoption of Kraichnan [72], is the most widely used method for creating synthetic turbulence in wind engineering applications. One of the earliest artificial turbulence techniques used in tall building applications was the RFG [46]. However, as mentioned by Huang et al. [73], the RFG is unable to replicate oscillations that create a PSD comparable to Von-Karman's, which is found in an atmospheric boundary layer (ABL).

An adaption of the works [71], [72] that imposes the Von-Karman spectrum in place of the Gaussian model is the Discretising Synthesising Random Flow Generator (DSRFG), which Huang et al. [73] presented in recognition of this shortcoming. Numerous models, including those by Castro et al. [74] and Kim et al. [75], which were additional modifications of the DSRFG, have been proposed in the literature. However, the Consistent Discrete-RFG (CDRFG) approach developed by Aboshosha et al. [76] has been widely employed in recent literature. Aboshosha et al. [76] rectified flaws in the target velocity spectrum and spatial coherency function of velocities in the CDRFG approach, which is a further adaption of the DSRFG. The CAARC building's alongwind, crosswind, and torsional moments are compared with wind tunnel data using the DSRFG and CDRFG.

Moment plots that are closest to the experimental findings may be represented by the CDRFG in particular. In the field of wind engineering, CDRFG has been widely accepted, and literature [77], [78] provides many instances of its usage on tall buildings. Additionally, the CDRFG approach has been applied to CWE applications such as topographical impacts [79], pedestrian level wind [79-81] and ABL development inquiry [52], [81-83]. New synthetic turbulence models based on the CDRFG have been published recently. Examples are Yu et al. [84] Narrow Band Synthesis RFG (NSRFG) and Patruno and Ricci's [83] Prescribed-wavevector RFG (PRFG) models. The goals of the PRFG and NSRFG synthetic turbulence models were to address the shortcomings of the CDRFG technique in terms of spatial length scale homogeneity and flow anisotropy.

At the domain's entrance, the majority of studies often assess profiles, velocity spectra, and spatial correlations [38]. The approach flow velocity data need to be measured at the building location, much like in an experimental wind tunnel investigation [45]. Dissipation, which is frequently caused by the capabilities of the synthetic model used, makes it challenging to sustain inflow conditions at the target point in CFD simulations [16], [45], [85]. Emphasis was laid on the significance of monitoring these parameters at the intake rather than in an empty domain at the intended building location. As the inflow gets closer to the intended construction location, inadequate mesh size can also have a big impact on the turbulent disturbances. This was made clear in the work by Wijesooriya et al. [66], where turbulent fluctuations were synthesized using the vortex approach.

The study demonstrated that the "flow's inflow characteristics are extremely sensitive to grid size and that adjusting the inflow characteristics requires many parametric experiments conducted in an empty domain. This suggests that to preserve the flow's inflow characteristics, some kind of mesh refinement are needed in the area between the inlet and the building placement. Therefore, while taking into account both inflow characteristics and building-induced velocity variations, it is necessary to strategically use mesh refinement and the placement of refined zones. The next part discusses mesh refining and domain size difficulties and how they directly affect simulation quality.

4 | Applications of CFD in tall Buildings and Structures

From the validation of isolated buildings to more complicated FSI in structural design applications to more advanced modelling, including aerodynamic shape change studies, CFD applications and research have advanced over time [19], [86-92].

4.1 | Tall building Validation Studies

The main focus of tall building validation research is on force coefficients and building surface pressures. Because experimental findings are available in the literature, many of the older articles primarily used the CAARC building to validate the CFD models against wind tunnel trials. The mean (CPM) and RMS of variations (the standard deviation) (CP, RMS) of surface pressures are frequently used for comparison. The pressure surrounding the CAARC building is often measured, compared, and displayed at two-thirds of the structure's height in the literature [93], [94].

Pressure measurements have also been investigated and verified for buildings with irregular geometry [95], [95], [96]. A tall structure with different circular cross-sections was examined by Mohotti et al. [97]. The purpose of the study was to assess how well RANS CFD models could forecast prominent flow characteristics found in circular cross-sections. According to Mohotti et al. [97], RANS turbulence models performed well in forecasting leeward pressure but poorly in forecasting windward and crosswind pressures. Although leeward pressure was improved by using the DDES model, it was still unsatisfactory. Subsequent research showed that,when comparing to the experimental results [66], the employment of LES modelling and polyhedral cells in the domain produced highly precise predictions for the same building [97].

Additionally, Yan and Li [98] investigated surface roughness and its impact on pressure predictions in an elliptical-shaped thin structure. LES was the most effective at forecasting pressure, drag, and lift coefficients among the scale-resolving turbulence models used. A study on high-rise rectangular buildings with and without balconies was carried out by Zheng et al. [99]. It was demonstrated that RANS models could only reliably forecast pressure coefficients that were windward; in contrast, LES models were necessary for situations when pressure was measured at other faces and when the angles of wind assault were not normal to the body's broad face. Last but not least, many scholars have examined a broad range of building forms and kinds, including E-shaped [100], L-shaped [101], and other random geometries [102]. The pressure distributions of these unusually shaped buildings, as well as the impacts of wake flow on downstream structures, are frequently covered in these studies.

4.2 | Aerodynamic Optimization

Designing external building forms and characteristics to minimize wind-induced stresses in both alongwind and crosswind orientations is known as aerodynamic optimization of buildings. As strong yet lightweight materials are used in current construction technologies to lower the structure's carbon footprint and overall construction cost, vibrations in buildings have become increasingly troublesome. Vortex-Induced Vibration (VIV), in which the frequency of crosswind shedding of vortices corresponds with the natural frequency of the structure, is the primary problem in contemporary tall structures. According to Irwin [103], one way to prevent a cross-wind reaction is to stiffen the building such that the peak resonance from VIV happens at considerably higher wind speeds, which are unlikely to occur within the building's design lifespan.

However, this would not be financially viable; thus, form optimization is the greatest way to prevent and minimize vortex shedding completely. The rapid modelling tools available have made CFD studies on form optimization a major field of research. Two-dimensional RANS modelling was used in some of the older investigations. One of the first experiments using a Kriging-based methodology was reported by Bernardini et al. [104], who claimed that it was considerably more effective than running random experimental models. By updating the Kriging models using the Pareto front framework, Bernardini et al. [104] achieved an optimal shape based on minimizing the drag and lift coefficients using just 0.75% of the anticipated CFD models.

Bernardini [104] employed a co-Kriging technique that combined LES and RANS modelling to assess an ideal cross-sectional shape; this was further improved by Ding & Kareem [105]. For aerodynamic optimization, Elshaer et al. [78] employed a surrogate model trained on 2D LES flow that was based on Artificial Neural Networks (ANNs). After determining the ideal aerodynamic shape, the results were confirmed in an ABL flow using a complete 3D LES model [106], [107]. It was demonstrated that the ideal solutions for lift and drag were reduced by 24% and 30%, respectively, using a square cross-section as the foundation model [107], [108].

Important concerns regarding aerodynamic shape alterations and their consequences on structures are covered in detail in a review conducted by Sharma et al. [109]. An Independent CFD study was also carried out by Sharma et al. [109] to demonstrate and confirm the efficacy of corner-modified, tapered, and set-backed building designs. From the standpoint of a structural engineer, form optimization is often accomplished to reduce drag and RMS lift forces for strong structural design. To guarantee that the best possible design is produced, the structural and wind engineers must collaborate closely with the architect. Despite being the most important design criterion, Serviceability Limit State (SLS) design, which considers building comfort in terms of occupant comfort, is often disregarded.

This is often constrained by the greatest deflection, but more significantly, by the maximum accelerations that the structure can withstand under SLS circumstances [108]. As a result, aerodynamic optimization is crucial to the structure's design, as Irwin [2] explained. One typical tactic used by designers to lessen wind-induced vibrations and stresses applied to the structure is the creation of apertures in the building [110]. According to Elshaer and Bitsuamlak [111], ANN-assisted LES simulations may be used to optimize opening positions on a building and lower total wind-induced forces by up to 47%. One disadvantage of traditional opening techniques is that they drastically diminish the amount of usable space and floor space that is accessible.

5 | Shielding Effects and City Modelling

The majority of the literature that is now accessible concentrates on isolated building scenarios due to early validation and basic research on wind flow behaviour around buildings. Nonetheless, it is well known that densely populated cities are typically home to towering skyscrapers [112]. Thus, it is also necessary to consider the adjacent structures and their impact. Practice codes like ASCE/SEI 49-21 [68] call for the adoption of "proximity models," which take into account nearby structures or topography that might negatively impact the subject structure's ability to withstand approach wind. As long as they are robust, proximity models may be constructed from foam and do not require complete architectural characteristics [68].

The extent of proximity models is generally hard to anticipate and depends on a case-by-case scenario that is assessed by the wind engineer, as stated in the code [68]. Significant structures are often included in the tunnel within a 300-1000 m radius (the exact distance varies depending on the tunnel's size) of the subject building [23]. In a study of upwind building interference, Dagnew and Bitsuamlak [85]contrasted an isolated building (Case 1) with an interference building half the height of the subject structure (Case 2) and an interference building the same height (Case 3). The intricate interference flow phenomenon was examined using the visual outputs after the numerical CFD LES model was verified against the experimental results. When comparing the half-height building (Case 2) to an isolated building scenario, it was found that all force coefficients were dropped.

In contrast, the subject structure in *Example (3)* was subject to greater RMS crosswind pressures. This was explained by CFD data, which showed that because of the subject structure's close vicinity, the flow of the interference building stayed separated (from vortex shedding). According to Dagnew and Bitsuamlak [85], this influence subsequently caused a broader wake in the subject structure. The interference effects of two tall buildings (with a 1:1:7 breadth, width, and height ratio) with different separation distances of 1.5-10 were also studied by Sharma et al. [113]. It was demonstrated that the interfering building had negative windward pressures at a tight separation distance (S = 1.5). This was comparable to the findings of Dagnew and Bitsuamlak [85], who also found that the affected building had negative windward pressures (S = 0.67).

Sharma et al. [113] found that buildings that are adjacent to one other in tandem act almost like a single huge body. The leeward face of the interfering building often experiences less negative pressure than the isolated structure because of its longer sides. Generally speaking, it has been demonstrated that the drag forces rise with increasing separation and the displacement of the interfered building from the interference building's wake until the two are so far away that careless shielding effects occur (S = 12 ∼ 16) [113]. Similar findings were also reported by Shirzadeh et al. [114], who investigated interference effects in which the interfering building is both upstream and downstream, staggered in tandem.

A single interference building is the focus of the majority of shielding effect investigations. However, in a research published by Elshaer et al. [115], they used LES CFD to first analyze the impacts of wind on a tall building subjected to conventional surrounding building obstacle configurations. The shielding effect on the interfering building rises with the height of the surrounding structure, as can be seen from the mean pressure coefficients. However, Elshaer et al. [115] observed that as the height of the surrounding structures increased, the alongwind and crosswind base moments on the interfering structure decreased, resulting in lower cumulative forces on the subject structure.

In summary, compared to an isolated building example, the presence of nearby buildings can often reduce the overall building pressures. The examination of the induced pressure on buildings caused by the expansion of Downtown Toronto between 1980 and 2015 is arguably the most fascinating aspect of the Elshaer et al. [115] study. While the majority of buildings experienced a general decrease in mean pressures, certain structures' mean pressure coefficients increased somewhat as a result of recent developments. This implies that negative consequences may arise when a city changes, particularly if the structure in issue and its façade components are outdated and require maintenance. Buildings may or may not be negatively impacted, but a changing city may raise concerns about wind comfort at the pedestrian level. Increased roughness causes the approach wind to become more turbulent. When combined with potential wind acceleration brought on by tunnelling processes, this turbulence can be a form of challenge for pedestrians [79], [115].

6 | Fluid-Structure Interaction

It is impossible to directly evaluate the aeroelastic reaction and its implications using the one-way FSI approach. Implementation and processing power, FSI modelling is quite difficult. Though they are significantly more demanding than one-way FSI simulations, two-way FSI simulations exhibit more realistic physical behaviour. The primary cause of this is the continuous feedback that occurs between the solvers throughout each time step. There isn't much literature accessible since FSI modelling is challenging and demanding, particularly when it comes to applying tall and tall structures.

Braun and Awruch [63] demonstrated one of the first FSI simulation implementations, using the CAARC building to run two-way FSI simulations. With a smooth inflow profile (no turbulence), an LES model was used. Their earlier work served as the foundation for the partitioned coupling system; interested readers are referred to Ref. [116]. In six simulations, Braun and Awruch [63] recorded the structure's reaction at six decreased velocities (Vr). The experimental findings of Thepmongkorn et al. [117], who employed a stick aero-elastic model to ascertain the building reaction, were contrasted with the numerical results. It was demonstrated that while the FSI simulation could properly depict alongwind deflection values, it did a poor job of estimating the cross-wind response.

Zhang et al. [118] suggested a one-way FSI analysis that only solved the fluid simulation once, pointing out that two-way FSI simulations were computationally taxing. This was accomplished by adjusting the natural frequency of the structures to determine the structural reaction at various decreased velocities. The mesh number (360 k) and the quantity of data outputs collected suggest that Braun and Awruch's [63] work may have required significant computer resources, even if this was not mentioned. On the other hand, Zhang et al. [118] reported that the structural simulation on 12 CPU cores took 35 minutes, while the CFD simulation on 96 CPU cores took 122 hours. Considering that greater numerical precision was achieved and that the results were equivalent to those of experiments, this is quite time-efficient.

The works of Hasama et al. [119] and Feng et al. [120] provide other attempts to use two-way FSI simulations. A rectangular prism's structural reaction was examined by Hasama et al. [119], who reduced the structure to a spring-mass system. For buildings that impede intricate structural components in the prototype building, this spring-mass system is perfect. When compared to the experimental results reported by Marukawa [121], it was demonstrated that the suggested approach could accurately estimate alongwind and crosswind responses. Nonetheless, Hasama et al. [119] observed some variations in the torsional responses and concluded that these variations could have been caused by structural simplification.

It is well known that complicated vibrational modes cannot be satisfactorily reproduced by spring and mass models. The study by Zheng et al. [122] demonstrated how a tall building's mass-spring system simplification resulted in differences for the fifth and sixth vibration modes that were greater than sixteen per cent. Simplifying the structural model is also counterintuitive, considering that representing the actual structural model of the prototype building is one of the primary benefits of using a numerical model to assess structural performance. In two sets of investigations, Feng et al. [120] first gave a validation test case for a structure that was 300 meters tall, and then they examined just numerical results for a building that was 1000 meters tall. Feng et al. [120] used the commercial code ANSYS [24] to perform the simulations and two-way coupling.

Because of the computational requirement, Feng et al. [120] only assessed the structures' performance at three Vr 's for the validation scenario. Additionally, the LES model used was imprecise, with an average y+ range from 15-70. Additionally, a condensed MDOF structural model was applied. The alongwind measurements (and, to a lesser degree, the crosswind) of the standard deviation response revealed differences in findings when compared to the experimental, as Feng et al. [120] explained. These are explained by Feng et al. [120] as a mix of variations seen in the physical model and incoming wind amounts. Although these arguments could be valid, inadequate mesh density is typically blamed for the actual discrepancy between experimental and numerical results.

The flow scales required to adequately represent the flow characteristics surrounding the structure cannot be recorded due to insufficient mesh density. The study also investigates the response of a 1000-meter-tall skyscraper, with and without aerodynamic damping, to wind-veering effects [120-122]. A multi-partitioned, fully linked FSI technique was proposed by Pentek et al. [123], which allowed for the inclusion of the structural model together with other devices such as mass dampers as part of the structural system. The paper presents intricate FSI simulations that account many parametric studies on the simplification of structural models and the inclusion of TMDs. Pentek et al. [123] only provided numerical results in their study, which employed the CAARC building to examine the first two basic sway modes of vibration.

In their innovative one-way FSI methodology, Wijesooriya et al. [124] presented a numerical framework for quickly obtaining wind-induced loads and reactions. The framework, which was designed for civil and structural applications, includes an ELES numerical modelling approach and a very effective pressure mapping technique that was initially presented in the author's prior article [96]. Zhang et al. [118] and the suggested approach in [96] used a comparable assessment methodology. To get the necessary measurement numbers at various Vr's, the structural model was modified and run many times after the CFD model was run once. The distinction was that an effective pressure mapping method was put forth, transferring the pressure load from CFD to mechanical through a script from MATLAB.

Pressure mapping from CFD to the mechanical solver was quick and extremely well-optimized. The implicit modal analysis used by the mechanical solver was likewise quite effective and produced answers in a matter of minutes [102]. Wijesooriya et al. [96] tested the CAARC construction in its initial implementation and demonstrated its performance in comparison to experimental results. An MDOF aeroelastic model was then evaluated in a wind tunnel for a tall building that was 300 meters high (408 meters with a stalk) [91]. When compared to the experimental results, the one-way FSI approach yielded high numerical accuracy in both alongwind and crosswind responses. Additionally, the building's acceleration and deflection control were optimized using the numerical model for occupant comfort [91].

Additionally, Wijesooriya et al. [94] used ANSYS [24] to compare the one-way FSI approach with a fully linked two-way FSI simulation that was comparable to the one used by Feng et al. [127]. The meshing and size of the CFD domain were varied, as was covered in [100]. A top-notch CFD mesh with 1.7 million polyhedral cells was supplied for the one-way FSI simulation. The two-way FSI simulation, on the other hand, employed a dynamic mesh (deformable and re-meshed as appropriate) with 1.6 million elements, hence the two-way FSI model had to use a hexahedral-tetrahedral mesh. Wijesooriya et al. [94] noted that a minimum of 3.2 million mesh elements were needed to forecast all flow aspects, while 1.6 million cells for the hexahedral-tetrahedral mesh were insufficient.

However, this compromise has to be made like that outlined in Feng et al. [120] because of the computational time limitation. a comparison of the crosswind and alongwind standard deviations of the tip deflection. As can be seen, the one-way FSI (u1w-FSI) curve has a high degree of accuracy in predicting deflection values. Due to its smaller data set, the two-way FSI (u2w-FSI) is unable to forecast significant resonance characteristics, as those found in peaks with the label "1," which indicate vortex-induced resonance for the building under investigation [94].

Six two-way FSI analyses were conducted to examine the computational requirement; only six data points were collected, and each one took 4800 clock hours to complete in an HPC [94]. The one-way FSI simulation, in contrast, only required 74 clock hours in total to provide simulation results for 18 distinct structure configurations. Two-way FSI simulations are complicated and demand a lot of processing resources. To get results in an acceptable amount of time, certain sacrifices must be made, including mesh density, cell type, minimum cell y+, and even overall flow duration. This indicates that the one-way FSI simulation produces better results, despite "the in principle" superior two-way FSI simulation's failure to provide better results. The one-way FSI model's incapacity to take into consideration negative aerodynamic damping, which often results from Vortex-Induced Resonance (VIR), is its sole drawback.

However, as noted in the author's [94] citation, VIR happens when velocities fall between 80 and 100 m/s for the majority of buildings with prismatic shapes, including squares and rectangles. Even ULS design speeds are exceeded by this. In summary, if improved flow characteristics and the atmospheric boundary layer replication can be simulated, which is accomplished using a one-way FSI modelling approach, then the tradeoff of excluding negative aerodynamic damping is justified. A one-way FSI study was also reported by Ricci et al. [125], in which the produced internal forces and the deflection of a tall structure were assessed and compared to wind tunnel tests with a strong correlation.

7 | Future of CFD Modelling and Other Promising Numerical Techniques on tall Structures

7.1 | The Lattice-Boltzmann method

Over the past ten years, computational technology has advanced significantly, and even the typical consumer may now easily access high-performance computers. Higher-order turbulence models, like LES, are now more practical and easily used in engineering applications as a result of these developments. Recently, the Lattice-Boltzmann Method (LBM) has become popular as a substitute numerical technique for CFD. Particle kinetic theory is used to simulate fluid flow physics in LBM, which has been shown to be a considerably more effective modelling method than the traditional CFD approach [126]. Studies like pedestrian-level wind of city models [127], [128] and pollutant dispersion , [129] are only two examples of the growing use of LBM-LES approaches for wind engineering.

Buffa et al. [130] conducted a numerical simulation using LBM-LES on a tall building, validating wind-induced loads and pressures as well as inflow characteristics. It was demonstrated that the LBM-LES approach could duplicate these values with a high degree of numerical precision. In a research on a high-rise structure, Han et al. [131] compared the LBM-LES approach with both an experimental and a traditional CFD (Finite Volume - FVM-LES). The parameters of fluctuation velocity, flow structures and reattachment length, TKE profiles, and averaged temporal velocity profiles were compared. It was demonstrated that LBM-LES could produce accurate predictions of experimental results with comparable precision to traditional FVM-LES modeling [131].

In order for the LBM-LES to capture the requisite flow characteristics, the study also focuses on the mesh refinement needs. An LBM-LES simulation over a high-rise building - the same structure that Han et al. [131] compared—was given by Santasmasas et al. [132]. Using LBM-LES in the region of interest, Santasmasas et al. [133] used a zonal strategy, which is quite similar to that used in Embedded-LES simulations [66]. It was demonstrated that for the structure's flow characteristics, the LBM-LES offered good agreement with experimental results. A wall-modelled LES (WMLES) simulation was used to compare these results to a traditional FVM-CFD simulation. It was demonstrated that, when compared to the FVM-CFD method, the most demanding LBM-LES simulation was still 25 times less expensive [132].

Although LBM is now mostly utilized as a research tool, it has a lot of promise for usage in real-world wind engineering applications in the future. LBM CFD is currently undergoing validation, particularly in Wind Engineering applications, much like the FVM approach did a few decades ago. Because collision theory and particle kinematics are used instead of direct conservation equation solution, which is what standard CFD code does, LBM's primary benefit is its quicker calculation time. Additionally, it scales effectively with CPUs and GPUs to carry out its functions and may leverage outstanding parallel performance with contemporary computer hardware [69].

The parallelization design seen in contemporary hardware has led to the widespread use of LBM on GPU architecture. One of LBM's primary disadvantages [132], is the need to retain a lot of data for solved values, which can occasionally significantly impair the performance of big simulations. This served as one of the primary drivers behind Santasmasas et al.'s implementation of Embedded-LES utilizing LBM [132]. Like FVM-CFD, LBM is still a developing technique that will become more mature and useful for a wider range of flow issues. Readers interested in learning more about the theoretical underpinnings of the LBM [64].

8 | Artificial Intelligence Approaches

The adoption of (AI)-driven methods in wind engineering has become more popular recently. The availability of enormous data sets gathered over years of experimental testing is mostly to blame for this. Salehi and Burgueno [134] provide an eloquent survey of the different engineering sectors that have used AI approaches, including structural health monitoring, systems identification, design, and optimization. Early research on (AI) [135], [136], particularly in wind engineering includes the work of Martínez-Vazquez & Rodríguez-Cuevas [137], who used an (ANN) technique to study wind field data reproduction.

Wu and Kareem [138] investigated the dynamic non-linearities of bridge decking using the ANN technique. Without the requirement for training data, Raissi et al. [139] demonstrated a Deep Learning (DL) system that can replicate crucial flow quantities of circular cylinders experiencing VIV, including the lift and drag forces. Moreover, velocity and pressure contours could be faithfully reproduced by the DL model [140]. Particularly when it comes to buildings, AI-driven methodologies are still in their infancy. A few recent research are described here, along with their results and potential applications.

Hu and Kwok [140] used Machine Learning (ML) to calculate the mean and RMS pressure distribution of circular structures, claiming that this method would be far more effective than CFD and wind tunnel approaches. They gathered information from 47 researches and compared predictions using the Decision Tree Regressor (DTR), Random Forest (RF), and Gradient Boosting Regression Tree (GBRT) ML techniques. Inflow velocity and turbulence have an impact on building surface pressures, particularly for circular cross-sections. As a result, the ML algorithms were trained to forecast pressure levels based on turbulence and Reynolds (Re) number. Hu and Kwok [140] demonstrated that ML algorithms, particularly the GBRT method, could produce these estimations with a respectable level of accuracy. An ML-based method was also used by Lin et al. [141] to calculate the cross-wind vibration displacements of rectangular cylinders.

The GBRT algorithm produced the best estimations out of all the ML algorithms that were examined [141]. In order to investigate pressure patterns on tall structures, Kim et al. [142] introduced an unsupervised ML approach that used a clustering technique. The significance of the observed pressure patterns was assessed using the clustering technique, which also helped to clarify the building's aerodynamic properties. ML techniques on crosswind load effects [143], wind surface pressure measurements [144], interference effects [145], and wind flow patterns on tall buildings [146] .Since AI algorithms can perform predictive measurements, critically analyze large datasets, and comprehend underlying patterns and causations of observed/acquired data, it is evident from the literature above that AI-driven approaches will play a significant role in wind engineering in the future.

While AI-driven methods are not as good as CFD-based numerical modeling, they are still numerical methods that can produce qualitative results, and their main advantage is that they can produce results at a reasonable and practical cost, especially when compared to wind tunnel methods. Additionally, AI-generated numerical outputs are far faster than numerical modeling based on CFD. Lastly, as more data is gathered, the dependability of AI-driven outputs will only increase. It will also be a great tool to supplement current techniques like wind tunnel testing and CFD-based numerical modeling of tall structures.

9 | Conclusions

RANS turbuen turbulence models were unable to produce accurate numerical predictions for geometries with complicated forms or curved geometrical characteristics. Because RANS turbulence models require less computing power, they were the primary focus of these early investigations. If precise pressure forecasts were needed, scale-resolving turbulence models were required, but this came at a price. All CFD simulations rely on turbulence modeling, but there are other significant modeling issues as well, such the creation of inflow quantities. It is also clear from actual testing that the atmospheric boundary layer flow must be accurately represented in the numerical domain. Although the book covered a variety of approaches, synthetic turbulence production techniques received particular attention because of their widespread use and minimal computational headroom.

Furthermore, geometries with complex shapes or curved geometrical properties could not be accurately predicted numerically by RANS turbulence models. RANS turbulence models were the main focus of these early studies since they demand less processing power. There was a cost associated with using scale-resolving turbulence models to provide accurate pressure projections. Turbulence modeling is the foundation of all CFD simulations, but there are other important modeling challenges as well, such the generation of inflow quantities. Actual testing also makes it evident that the numerical domain needs to appropriately depict the atmospheric boundary layer movement. Even though the book discussed many different methods, synthetic turbulence creation techniques were given special consideration due to their extensive application and little computational headroom.

Many researchers have developed synthetic turbulence models, including the DSRFG [147], CDRFG [76], and NSRFG [84] techniques, which were specifically created to replicate atmospheric boundary layer flow and its uses for building design. As the atmospheric boundary layer representation in the CFD domain was accomplished, these studies played a crucial role in the development of CFD techniques on tall buildings. The methods for mesh generation and domain development are responsible for another facet of CFD modelling. Given that a high-quality grid must be able to resolve all required scales of flow for the given problem, the latter is perhaps more significant. Tetrahedral, hexahedral, and polyhedral forms were among the cell types and grid generation approaches that were covered.

According to recent research, polyhedral cells may produce high-quality mesh with a significantly lower cell count than hexahedral cells [92]. Complex geometric constructions where hexahedral meshing is challenging to accomplish are best suited for polyhedral meshing. With the help of the author's earlier work, the significance of refinement areas, including local mesh sizes on the subject wall, was examined [97], [66]. When using LES-type models, the y+ criterion is crucial, and the paper offers a way to get a preliminary estimate using computations and a few condensed RANS simulations. There was discussion and presentation of the consequences of improperly recording the wall boundary layer flow on curved surfaces.

Four key issues were then used to describe these modelling approaches and their applicability based on validation work done over the years by different academics. Due to the availability of experimental findings, early research that concentrated on pressure validation tests revealed that the CAARC building was the most well-liked by researchers. The intricacies and consequences of how input quantities may significantly impact pressure distribution, particularly for RMS quantities, were demonstrated by cross-examination of many numerical data from different studies (for the CAARC building). It was emphasized how crucial it is to measure inflow amounts in an empty domain at the building location. Additionally, structures with irregular geometries, varying cross-sectional forms, and tapered (or setback) profiles were examined, demonstrating the effectiveness of CFD.

Naturally, the conversation influenced the development of optimization techniques and their use in CFD. There was the discussion of Artificial Intelligence (AI) and algorithm-based frameworks for creating ideal forms, most of which used 2D modelling methods. Additionally, research on city modelling, shielding effects, and the efficiency of a CFD technique was covered. CFD was appropriate for optimization studies because it could measure and collect data without being constrained by instruments and easily changing buildings and surrounding elements. This publication placed a great deal of attention on FSI simulations because very few previous review articles by other scholars [38], [48-50] have thoroughly discussed this topic.

Utilizing the whole structural model of the prototype model is one advantage of employing a computational approach to numerical aeroelastic modelling. An MDOF model of restricted vibration modes is used in experimental aeroelastic models. For certain systems with complicated vibrational modes (particularly coupled ones), it might be challenging to recreate an experimental model. Furthermore, it takes time and experience to create such a model. A complete aero-elastic model is typically saved for the latter end of the building design, as Irwin et al. [103] noted. On the other hand, since creating a complete FE model for structural analysis is frequently standard procedure, the entire prototype building is typically accessible numerically.

The two categories of FSI modelling coupled (two-way) and uncoupled (one-way) simulations were explored. Although it is more difficult to execute, the former is a more exact physical portrayal. Due to the computing demands of two-way FSI, mesh quality and statistics must be sacrificed to get results in a reasonable amount of time. It was demonstrated that the majority of research employs one-way FSI methodologies to get accurate results comparable to those of aeroelastic experimental testing, which inevitably resulted in worse overall outcomes. The novel one-way FSI strategies put out by the authors [91], [94] appear to be the most practical choice available at the moment.

The study concludes by outlining potential future modelling opportunities for CFD, including the Lattice-Boltzmann Method (LBM) and further numerical methods utilizing AI. A synopsis of the two LBMs and AI, as well as their possible applications in wind engineering simulations for tall building design, were covered in this part. According to preliminary research and current literature, both approaches show great promise and might soon be completely implemented in real-world applications. Numerical approaches are undoubtedly here to stay and are constantly becoming better. The main design tool for wind engineering tall building design has historically been experimental approaches, which may be supported by these numerical tools to better optimize designs.

Author contributions

AAB: Conceptualization, Writing - original draft, Supervision; MOA: Writing - original draft, Introduction, Discussions; MAS: Resources, Writing; The authors read and approved the final manuscript.

Data availability

The study has no associated data.

Funding

This research received no external funding.

Acknowledgement

The authors appreciate the editor and reviewers for adding valuable input to the manuscript.

Conflicts of Interest

The author declares that there is no competing interest.

References

[1] Lamb, S., Kwok, K. C. S., & Walton, D. (2013). Occupant comfort in wind-excited tall buildings: Motion sickness, compensatory behaviours and complaint. *Journal of wind engineering and industrial aerodynamics*, *119*, 1–12. https://doi.org/10.1016/j.jweia.2013.05.004

[2] Irwin, P. A. (2009). Wind engineering challenges of the new generation of super-tall buildings. *Journal of wind engineering and industrial aerodynamics*, *97*(7–8), 328–334. https://doi.org/10.1016/j.jweia.2009.05.001

[3] Jensen, M. (1958). The model law for phenomena in natural wind. *Reprint from ingenioren (international edition)*, *2*(4), 121–128. https://cir.nii.ac.jp/crid/1573105974420639360

[4] Cochran, L., & Derickson, R. (2011). A physical modeler's view of computational wind engineering. *Journal of wind engineering and industrial aerodynamics*, *99*(4), 139–153. https://doi.org/10.1016/j.jweia.2011.01.015

[5] Tschanz, T., & Davenport, A. G. (1983). The base balance technique for the determination of dynamic wind loads. *Journal of wind engineering and industrial aerodynamics*, *13*(1), 429–439. https://doi.org/10.1016/0167-6105(83)90162-9

[6] Xie, J., & Irwin, P. A. (1998). Application of the force balance technique to a building complex. *Journal of wind engineering and industrial aerodynamics*, *77–78*, 579–590. https://doi.org/10.1016/S0167-6105(98)00174-3

[7] Pozzuoli, C., Bartoli, G., Peil, U., & Clobes, M. (2013). Serviceability wind risk assessment of tall buildings including aeroelastic effects. *Journal of wind engineering and industrial aerodynamics*, *123*. http://dx.doi.org/10.1016/j.jweia.2013.09.014

[8] Huang, M. F., Tse, K. T., Chan, C. M., Kwok, K. C. S., Hitchcock, P. A., Lou, W. J., & Li, G. (2010). An integrated design technique of advanced linear-mode-shape method and serviceability drift optimization for tall buildings with lateral–torsional modes. *Engineering structures*, *32*(8), 2146–2156. https://doi.org/10.1016/j.engstruct.2010.03.017

[9] Holmes, J. D. (1987). Mode shape corrections for dynamic response to wind. *Engineering structures*, *9*(3), 210–212. https://doi.org/10.1016/0141-0296(87)90017-4

[10] Boggs, D., & Peterka, J. (1989). Aerodynamic model tests of tall buildings. *Journal of engineering mechanics-asce*, *115*. http://dx.doi.org/10.1061/(ASCE)0733-9399(1989)115:3(618)

[11] Srivastava, K. N., & Raj, R. (2022). Aerodynamic study of tall buildings under wind load. *International journal of health sciences*, (III), 2620–2641. https://doi.org/10.53730/ijhs.v6nS3.6142

[12] Tse, K. T., Hitchcock, P. A., & Kwok, K. C. S. (2009). Mode shape linearization for HFBB analysis of wind-excited complex tall buildings. *Engineering structures*, *31*(3), 675–685. https://doi.org/10.1016/j.engstruct.2008.11.012

[13] Zou, L., Shi, T., Song, J., & Cai, C. S. (2017). Application of the high-frequency base balance technique to tall slender structures considering the effects of higher modes. *Engineering structures*, *151*, 1–10. https://doi.org/10.1016/j.engstruct.2017.08.005

[14] Cui, W., & Caracoglia, L. (2016). Physics-based method for the removal of spurious resonant frequencies in high-frequency force balance tests. *Journal of structural engineering*, *142*(2), 4015129. https://doi.org/10.1061/(ASCE)ST.1943-541X.0001414

[15] Chen, X., & Kareem, A. (2005). Dynamic wind effects on buildings with 3D coupled modes: Application of high frequency force balance measurements. *Journal of engineering mechanics*, *131*(11), 1115–1125. https://doi.org/10.1061/(ASCE)0733-9399(2005)131:11(1115)

[16] Alkhatib, F., Kasim, N., Goh, W. I., Shafiq, N., Amran, M., Kotov, E. V., & Albaom, M. A. (2022). Computational aerodynamic optimization of wind-sensitive irregular tall buildings. *Buildings*, *12*(7), 939. https://doi.org/10.3390/buildings12070939

[17] Rosa, L., Tomasini, G., Zasso, A., & Aly, A. M. (2012). Wind-induced dynamics and loads in a prismatic slender building: A modal approach based on unsteady pressure measurements. *Journal of wind engineering and industrial aerodynamics*, *107/108*, 118–130. https://doi.org/10.1016/j.jweia.2012.03.034

[18] Aly, A. M. (2013). Pressure integration technique for predicting wind-induced response in high-rise buildings. *Alexandria engineering journal*, *52*(4), 717–731. https://doi.org/10.1016/j.aej.2013.08.006

[19] Assainar, N., & Dalui, S. K. (2021). Aerodynamic analysis of pentagon-shaped tall buildings. *Asian journal of civil engineering*, *22*(1), 33–48. https://doi.org/10.1061/9780784412909.083

[20] Dongmei, H., Ledong, Z., Quanshun, D., Xue, Z., & Wei, C. (2017). Aeroelastic and aerodynamic interference effects on a high-rise building. *Journal of fluids and structures*, *69*, 355–381. DOI:https://doi.org/10.1016/j.jfluidstructs.2017.01.007

[21] Wang, L., Fan, X.-Y., Liang, S.-G., Song, J., & Wang, Z.-K. (2018). Improved expression for across-wind aerodynamic damping ratios of super high-rise buildings. *Journal of wind engineering and industrial aerodynamics*, *176*, 263–272. https://doi.org/10.1016/j.jweia.2018.04.001

[22] Zheng, C., Liu, Z., Wu, T., Wang, H., Wu, Y., & Shi, X. (2019). Experimental investigation of vortex-induced vibration of a thousand-meter-scale mega-tall building. *Journal of fluids and structures*, *85*, 94–109. https://doi.org/10.1016/j.jfluidstructs.2018.12.005

[23] Hou, F., & Sarkar, P. (2021). Time-domain model for prediction of generalized 3DOF buffeting response of tall buildings using 2D aerodynamic sectional properties. *Engineering structures*, *232*, 111847. https://doi.org/10.1016/j.engstruct.2020.111847

[24] Solari, G. (2017). Wind loading of structures: framework, phenomena, tools and codification. *Structures*, *12*, 265–285. https://doi.org/10.1016/j.istruc.2017.09.008

[25] Zhou, Y., & Kareem, A. (2003). Aeroelastic balance. *Journal of engineering mechanics*, *129*(3), 283–292. https://doi.org/10.1061/(ASCE)0733-9399(2003)129:3(283)

[26] Blocken, B. (2011). Computational Wind engineering: theory and applications. In *Environmental wind engineering and design of wind energy structures* (pp. 55–93). Springer. https://link.springer.com/chapter/10.1007/978-3-7091-0953-3_3

[27] Blocken, B. (2014). 50 years of computational wind engineering: past, present and future. *Journal of wind engineering and industrial aerodynamics*, *129*, 69–102. https://doi.org/10.1016/j.jweia.2014.03.008

[28] Blocken, B., Stathopoulos, T., & Van Beeck, J. (2016). Pedestrian-level wind conditions around buildings: Review of wind-tunnel and CFD techniques and their accuracy for wind comfort assessment. *Building and environment*, *100*, 50–81. https://doi.org/10.1016/j.buildenv.2016.02.004

[29] Alizadeh, M., & Sadrameli, S. M. (2018). Numerical modeling and optimization of thermal comfort in building: Central composite design and CFD simulation. *Energy and buildings*, *164*, 187–202. https://doi.org/10.1016/j.enbuild.2018.01.006

[30] Shan, X., Luo, N., Sun, K., Hong, T., Lee, Y. K., & Lu, W. Z. (2020). Coupling CFD and building energy modelling to optimize the operation of a large open office space for occupant comfort. *Sustainable cities and society*, *60*, 102257. https://doi.org/10.1016/j.scs.2020.102257

[31] Rodi, W. (1993). On the simulation of turbulent flow past bluff bodies. *Journal of wind engineering and industrial aerodynamics*, *46*, 3–19. https://doi.org/10.1016/0167-6105(93)90111-Z

[32] Murakami, S. (1997). Current status and future trends in computational wind engineering. *Journal of wind engineering and industrial aerodynamics*, *67*, 3–34. https://doi.org/10.1016/S0167-6105(97)00230-4

[33] Lübcke, H., Rung, T., & Thiele, F. (2001). Comparison of LES and RANS in bluff-body flows. *Journal of wind engineering and industrial aerodynamics*, *89*(14–15), 1471–1485. https://doi.org/10.1016/S0167-6105(01)00134-9

[34] Rodi, W. (1997). Comparison of LES and RANS calculations of the flow around bluff bodies. *Journal of wind engineering and industrial aerodynamics*, *69*, 55–75. https://doi.org/10.1016/S0167-6105(97)00147-5

[35] Kravchenko, A. G., & Moin, P. (2000). Numerical studies of flow over a circular cylinder at Re D= 3900. *Physics of fluids*, *12*(2), 403–417. https://dx.doi.org/10.1063/1.870318

[36] Wilcox, D. C. (1988). Reassessment of the scale-determining equation for advanced turbulence models. *AIAA journal*, *26*(11), 1299–1310. https://doi.org/10.2514/3.10041

[37] Menter, F. R. (1994). Two-equation eddy-viscosity turbulence models for engineering applications. *AIAA journal*, *32*(8), 1598–1605. https://doi.org/10.2514/3.12149

[38] Jafari, M., & Alipour, A. (2021). Methodologies to mitigate wind-induced vibration of tall buildings: A state-of-the-art review. *Journal of building engineering*, *33*, 101582. https://doi.org/10.1016/j.jobe.2020.101582

[39] Tominaga, Y., & Stathopoulos, T. (2013). CFD simulation of near-field pollutant dispersion in the urban environment: A review of current modeling techniques. *Atmospheric environment*, *79*, 716–730. https://doi.org/10.1016/j.atmosenv.2013.07.028

[40] Spalart, P. R. (1997). *Comments on the feasibility of les for wings and on the hybrid rans/les approach* [presentation]. Proceedings of the first afosr international conference on dns/les, 1997 (pp. 137–147). https://doi.org/10.1115/GT2010-22394

[41] Spalart, P. R., Deck, S., Shur, M. L., Squires, K. D., Strelets, M. K., & Travin, A. (2006). A new version of detached-eddy simulation, resistant to ambiguous grid densities. *Theoretical and computational fluid dynamics*, *20*, 181–195. https://doi.org/10.1007/s00162-006-0015-0%0A%0A

[42] Menter, F. R., Kuntz, M., Langtry, R., & others. (2003). Ten years of industrial experience with the SST turbulence model. *Turbulence, heat and mass transfer*, *4*(1), 625–632. https://doi.org/10.1016/S0167-6105(01)00134-9

[43] Mochida, A., Murakami, S., Shoji, M., & Ishida, Y. (1993). Numerical simulation of flowfield around Texas tech building by large eddy simulation. In *Computational wind engineering 1* (pp. 455–460). Oxford: Elsevier. https://doi.org/10.1016/B978-0-444-81688-7.50050-4

[44] Panneer Selvam, R. (1996). Computation of flow around Texas Tech building using k-ϵ and Kato-Launder k-ϵ turbulence model. *Engineering structures*, *18*(11), 856–860. https://doi.org/10.1016/0141-0296(95)00156-5

[45] Hui, Y. W., Al-Obaidi, A. S. M., Mari, T. S., Gunasagaran, S., & Ching, M. (2023). Investigation of the effect of the wind speed on the aerodynamic and architectural design of tall buildings. *18th international engineering research conference* (Vol. 2523, p. 12039). IOP Publishing. http://dx.doi.org/10.1088/1742-6596/2523/1/012039

[46] Huang, S., Li, Q. S., & Xu, S. (2007). Numerical evaluation of wind effects on a tall steel building by CFD. *Journal of constructional steel research*, *63*(5), 612–627. https://doi.org/10.1016/j.jcsr.2006.06.033

[47] Tanaka, H., & Lawen, N. (1986). Test on the CAARC standard tall building model with a length scale of 1 : 1000. *Journal of wind engineering and industrial aerodynamics*, *25*(1), 15–29. https://doi.org/10.1016/0167-6105(86)90102-9

[48] Hou, F., & Jafari, M. (2020). Investigation approaches to quantify wind-induced load and response of tall buildings: A review. *Sustainable cities and society*, *62*, 102376. https://doi.org/10.1016/j.scs.2020.102376

[49] Abu-Zidan, Y., Mendis, P., Gunawardena, T., Mohotti, D., & Fernando, S. (2022). Wind design of tall buildings: The state of the art. *Electronic journal of structural engineering*, *22*(01), 53–71. https://doi.org/10.56748/ejse.2233101

[50] Thordal, M. S., Bennetsen, J. C., & Koss, H. H. H. (2019). Review for practical application of CFD for the determination of wind load on high-rise buildings. *Journal of wind engineering and industrial aerodynamics*, *186*, 155–168. https://doi.org/10.1016/j.jweia.2018.12.019

[51] Blazek, J. (2015). *Computational fluid dynamics: principles and applications*. Butterworth-Heinemann.

[52] Wang, X., Wen, G., & Wei, Y. (2024). Aerodynamic Noise Simulation of a Super-High-Rise Building Facade with Shark-Like Grooved Skin. *Biomimetics*, *9*(9). https://www.mdpi.com/2313-7673/9/9/570

[53] Diaz-Daniel, C., Laizet, S., & Vassilicos, J. C. (2017). Direct numerical simulations of a wall-attached cube immersed in laminar and turbulent boundary layers. *International journal of heat and fluid flow*, *68*, 269–280. https://doi.org/10.1016/j.ijheatfluidflow.2017.09.015

[54] Chen, W., Ji, C., Xu, D., & Zhang, Z. (2022). Three-dimensional direct numerical simulations of vortex-induced vibrations of a circular cylinder in proximity to a stationary wall. *Physics review fluids*, *7*(4), 44607. DOI:10.1103/PhysRevFluids.7.044607

[55] Chen, W., Ji, C., Xu, D., & Alam, M. M. (2022). Three-dimensional direct numerical simulations of two interfering side-by-side circular cylinders at intermediate spacing ratios. *Applied ocean research*, *123*, 103162. https://doi.org/10.1016/j.apor.2022.103162

[56] Zhao, M., Mamoon, A.-A., & Wu, H. (2021). Numerical study of the flow past two wall-mounted finite-length square cylinders in tandem arrangement. *Physics of fluids*, *33*(9). https://doi.org/10.1063/5.0058394

[57] Tominaga, Y. (2015). Flow around a high-rise building using steady and unsteady RANS CFD: Effect of large-scale fluctuations on the velocity statistics. *Journal of wind engineering and industrial aerodynamics*, *142*, 93–103. https://doi.org/10.1016/j.jweia.2015.03.013

[58] Smagorinsky, J. (1963). General circulation experiments with the primitive equations: I. The basic experiment. *Monthly weather review*, *91*(3), 99–164. https://doi.org/10.1175/1520-0493(1963)091%3C0099:GCEWTP%3E2.3.CO;2

[59] Germano, M., Piomelli, U., Moin, P., & Cabot, W. H. (1991). A dynamic subgrid-scale eddy viscosity model. *Physics of fluids a: Fluid dynamics*, *3*(7), 1760–1765. https://doi.org/10.1063/1.857955

[60] Lilly, D. K. (1992). A proposed modification of the germano sugrid-scale closure method. *Phys fluids a*, *4*, 633–635. https://doi.org/10.1063/1.858280

[61] Kim, W.-W., Menon, S., Kim, W.-W., & Menon, S. (1997). *Application of the localized dynamic subgrid-scale model to turbulent wall-bounded flows* [presentation]. 35th aerospace sciences meeting and exhibit (p. 210). https://doi.org/10.2514/6.1997-210

[62] Murakami, S. (1993). Comparison of various turbulence models applied to a bluff body. In *Computational wind engineering 1* (pp. 21–36). Elsevier. https://doi.org/10.1016/B978-0-444-81688-7.50006-1

[63] Braun, A. L., & Awruch, A. M. (2009). Aerodynamic and aeroelastic analyses on the CAARC standard tall building model using numerical simulation. *Computers & structures*, *87*(9–10), 564–581. https://doi.org/10.1016/j.compstruc.2009.02.002

[64] Yan, B. W., & Li, Q. S. (2015). Inflow turbulence generation methods with large eddy simulation for wind effects on tall buildings. *Computers & fluids*, *116*, 158–175. https://doi.org/10.1016/j.compfluid.2015.04.020

[65] Murakami, S. (1998). Overview of turbulence models applied in CWE--1997. *Journal of wind engineering and industrial aerodynamics*, *74*, 1–24. https://doi.org/10.1016/S0167-6105(98)00004-X

[66] Wijesooriya, K., Mohotti, D., Chauhan, K., & Dias-da-Costa, D. (2019). Numerical investigation of scale resolved turbulence models (LES, ELES and DDES) in the assessment of wind effects on supertall structures. *Journal of building engineering*, *25*, 100842. https://doi.org/10.1016/j.jobe.2019.100842

[67] Obasaju, E. D. (1992). Measurement of forces and base overturning moments on the CAARC tall building model in a simulated atmospheric boundary layer. *Journal of wind engineering and industrial aerodynamics*, *40*(2), 103–126. https://doi.org/10.1016/0167-6105(92)90361-D

[68] of Civil Engineers, A. S. (2022). *Wind tunnel testing for buildings and other structures* [presentation]. https://doi.org/10.1061/9780784415740

[69] Lenz, S., Schönherr, M., Geier, M., Krafczyk, M., Pasquali, A., Christen, A., & Giometto, M. (2019). Towards real-time simulation of turbulent air flow over a resolved urban canopy using the cumulant lattice Boltzmann method on a GPGPU. *Journal of wind engineering and industrial aerodynamics*, *189*, 151–162. https://doi.org/10.1016/j.jweia.2019.03.012

[70] Thordal, M. S., Bennetsen, J. C., Capra, S., & Koss, H. H. H. (2020). Engineering approach for a CFD inflow condition using the precursor database method. *Journal of wind engineering and industrial aerodynamics*, *203*, 104210. https://doi.org/10.1016/j.jweia.2020.104210

[71] Smirnov, A., Shi, S., & Celik, I. (2001). Random flow generation technique for large eddy simulations and particle-dynamics modeling. *123*(2), 359–371. https://doi.org/10.1115/1.1369598

[72] Kraichnan, R. H. (1970). Diffusion by a random velocity field. *The physics of fluids*, *13*(1), 22–31. https://doi.org/10.1063/1.1692799

[73] Huang, S. H., Li, Q. S., & Wu, J. R. (2010). S.H. Huang, Q.S. Li, J.R. Wu, A general inflow turbulence generator for large eddy simulation. *Journal of wind engineering and industrial aerodynamics*, *98*(10–11), 600–617. https://doi.org/10.1016/j.jweia.2010.06.002

[74] Castro, H. G., Paz, R. R., & Sonzogni, V. E. (2011). Generation of turbulent inlet velocity conditions for large eddy simulations. *Mecánica computacional*, *30*(29), 2275–2288. https://www.researchgate.net/publication/283362091_Generation_of_Turbulence_Inlet_Velocity_Conditions_for_Large_Eddy_Simulation

[75] Kim, Y., Castro, I. P., & Xie, Z.-T. (2013). Divergence-free turbulence inflow conditions for large-eddy simulations with incompressible flow solvers. *Computers & fluids*, *84*, 56–68. https://doi.org/10.1016/j.compfluid.2013.06.001

[76] Aboshosha, H., Elshaer, A., Bitsuamlak, G. T., & El Damatty, A. (2015). Consistent inflow turbulence generator for LES evaluation of wind-induced responses for tall buildings. *Journal of wind engineering and industrial aerodynamics*, *142*, 198–216. https://doi.org/10.1016/j.jweia.2015.04.004

[77] Elshaer, A., Aboshosha, H., Bitsuamlak, G., El Damatty, A., & Dagnew, A. (2016). LES evaluation of wind-induced responses for an isolated and a surrounded tall building. *Engineering structures*, *115*, 179–195. https://doi.org/10.1016/j.engstruct.2016.02.026

[78] Elshaer, A., Bitsuamlak, G., & El Damatty, A. (2017). Enhancing wind performance of tall buildings using corner aerodynamic optimization. *Engineering structures*, *136*, 133–148. https://doi.org/10.1016/j.engstruct.2017.01.019

[79] Adamek, K., Vasan, N., Elshaer, A., English, E., & Bitsuamlak, G. (2017). Pedestrian level wind assessment through city development: A study of the financial district in Toronto. *Sustainable cities and society*, *35*, 178–190. https://doi.org/10.1016/j.scs.2017.06.004

[80] Aliabadi, A. A., Veriotes, N., & Pedro, G. (2018). A Very Large-Eddy Simulation model for the investigation of the neutral atmospheric boundary layer. *Journal of wind engineering and industrial aerodynamics*, *183*, 152–171. https://doi.org/10.1016/j.jweia.2018.10.014

[81] Yang, Q., Zhou, T., Yan, B., Liu, M., Van Phuc, P., & Shu, Z. (2021). LES study of topographical effects of simplified 3D hills with different slopes on ABL flows considering terrain exposure conditions. *Journal of wind engineering and industrial aerodynamics*, *210*, 104513. https://doi.org/10.1016/j.jweia.2020.104513

[82] Um, J., & Han, N. (2021). Understanding the relationships between global supply chain risk and supply chain resilience: the role of mitigating strategies. *Supply chain management: an international journal*, *26*(2), 240–255. https://doi.org/10.1108/SCM-06-2020-0248

[83] Patruno, L., & Ricci, M. (2017). On the generation of synthetic divergence-free homogeneous anisotropic turbulence. *Computer methods in applied mechanics and engineering*, *315*, 396–417. https://doi.org/10.1016/j.cma.2016.11.005

[84] Yu, Y., Yang, Y., & Xie, Z. (2018). A new inflow turbulence generator for large eddy simulation evaluation of wind effects on a standard high-rise building. *Building and environment*, *138*, 300–313. https://doi.org/10.1016/j.buildenv.2018.03.059

[85] Dagnew, A. K., & Bitsuamlak, G. T. (2014). Computational evaluation of wind loads on a standard tall building using LES. *Wind and structures*, *18*(5), 567–598. https://doi.org/10.12989/was.2014.18.5.567

[86] Tominaga, Y., Mochida, A., Yoshie, R., Kataoka, H., Nozu, T., Yoshikawa, M., & Shirasawa, T. (2008). AIJ guidelines for practical applications of CFD to pedestrian wind environment around buildings. *Journal of wind engineering and industrial aerodynamics*, *96*(10–11), 1749–1761. https://doi.org/10.1016/j.jweia.2008.02.058

[87] Blocken, B. (2015). Computational fluid dynamics for urban physics: Importance, scales, possibilities, limitations and ten tips and tricks towards accurate and reliable simulations. *Building and environment*, *91*, 219–245. https://doi.org/10.1016/j.buildenv.2015.02.015

[88] Blocken, B., Stathopoulos, T., & Carmeliet, J. (2007). CFD simulation of the atmospheric boundary layer: wall function problems. *Atmospheric environment*, *41*(2), 238–252. https://doi.org/10.1016/j.atmosenv.2006.08.019

[89] Franke, J., Hirsch, C., Jensen, A., Krüs, H., Schatzmann, M., Westbury, P., … & Wright, N. (2004). Recommendations on the use of CFD in wind engineering. *Proceedings of the international conference on urban wind engineering and building aerodynamics*.

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

[91] Wijesooriya, K., Mohotti, D., Amin, A., & Chauhan, K. (2021). Wind loads on a super-tall slender structure: A validation of an uncoupled fluid-structure interaction (FSI) analysis. *Journal of building engineering*, *35*, 102028. https://doi.org/10.1016/j.jobe.2020.102028

[92] Peric, M., & Ferguson, S. (2005). The advantage of polyhedral meshes. *Dynamics*, *24*(45), 504. https://B2n.ir/pz8449

[93] Chandra, P., & Fisher, M. L. (1994). Coordination of production and distribution planning. *European journal of operational research*, *72*(3), 503–517. https://doi.org/10.1016/0377-2217(94)90419-7

[94] Wijesooriya, K., Mohotti, D., Amin, A., & Chauhan, K. (2021). Comparison between an uncoupled one-way and two-way fluid structure interaction simulation on a super-tall slender structure. *Engineering structures*, *229*, 111636. https://doi.org/10.1016/j.engstruct.2020.111636

[95] Daniels, S. J., Castro, I. P., & Xie, Z. T. (2013). Peak loading and surface pressure fluctuations of a tall model building. *Journal of wind engineering and industrial aerodynamics*, *120*, 19–28. https://doi.org/10.1016/j.jweia.2013.06.014

[96] Wijesooriya, K., Mohotti, D., Amin, A., & Chauhan, K. (2020). An uncoupled fluid structure interaction method in the assessment of structural responses of tall buildings. *Structures*, *25* 448-462. https://doi.org/10.1016/j.istruc.2020.03.031

[97] Mohotti, D., Wijesooriya, K., & Dias-da-Costa, D. (2019). Comparison of reynolds averaging navier-stokes (RANS) turbulent models in predicting wind pressure on tall buildings. *Journal of building engineering*, *21*, 1–17. https://doi.org/10.1016/j.jobe.2018.09.021

[98] Yan, B. W., & Li, Q. S. (2017). Detached-eddy and large-eddy simulations of wind effects on a high-rise structure. *Computers & fluids*, *150*, 74–83. https://doi.org/10.1016/j.compfluid.2017.02.009

[99] Zheng, X., Montazeri, H., & Blocken, B. (2020). CFD simulations of wind flow and mean surface pressure for buildings with balconies: Comparison of RANS and LES. *Building and environment*, *173*, 106747. https://doi.org/10.1016/j.buildenv.2020.106747

[100] Bhattacharyya, B., & Dalui, S. K. (2020). Experimental and numerical study of wind-pressure distribution on irregular-plan-shaped building. *Journal of structural engineering*, *146*(7), 4020137. https://doi.org/10.1061/(ASCE)ST.1943-541X.0002686

[101] Yuan, W., Wang, Z., Chen, H., & Fan, K. (2017). Numerical analyses of aerodynamic characteristics of integrated L-shaped high-rise building. *Advances in engineering software*, *114*, 144–153. https://doi.org/10.1016/j.advengsoft.2017.06.018

[102] Sanyal, P., & Dalui, S. K. (2021). Effects of internal angle between limbs of "Y" plan shaped tall building under wind load. *Journal of building engineering*, *33*, 101843. https://doi.org/10.1016/j.jobe.2020.101843

[103] Irwin, P., Kilpatrick, J., Robinson, J., & Frisque, A. (2008). Wind and tall buildings: negatives and positives. *The structural design of tall and special buildings*, *17*(5), 915–928. https://doi.org/10.1002/tal.482

[104] Bernardini, E., Spence, S. M. J., Wei, D., & Kareem, A. (2015). Aerodynamic shape optimization of civil structures: A CFD-enabled Kriging-based approach. *Journal of wind engineering and industrial aerodynamics*, *144*, 154–164. https://doi.org/10.1016/j.jweia.2015.03.011

[105] Ding, F., & Kareem, A. (2018). A multi-fidelity shape optimization via surrogate modeling for civil structures. *Journal of wind engineering and industrial aerodynamics*, *178*, 49–56. https://doi.org/10.1016/j.jweia.2018.04.022

[106] Alminhana, G. W., Braun, A. L., & Loredo-Souza, A. M. (2018). A numerical-experimental investigation on the aerodynamic performance of CAARC building models with geometric modifications. *Journal of wind engineering and industrial aerodynamics*, *180*, 34–48. https://doi.org/10.1016/j.jweia.2018.07.001

[107] Tanaka, H., Tamura, Y., Ohtake, K., Nakai, M., & Kim, Y. C. (2012). Experimental investigation of aerodynamic forces and wind pressures acting on tall buildings with various unconventional

configurations. *Journal of wind engineering and industrial aerodynamics*, *107*, 179–191. https://doi.org/10.1016/j.jweia.2012.04.014

[108] Johann, F. A., Carlos, M. E. N., & Ricardo, F. L. S. (2015). Wind-induced motion on tall buildings: A comfort criteria overview. *Journal of wind engineering and industrial aerodynamics*, *142*, 26–42. https://doi.org/10.1016/j.jweia.2015.03.001

[109] Sharma, A., Mittal, H., & Gairola, A. (2018). Mitigation of wind load on tall buildings through aerodynamic modifications. *Journal of building engineering*, *18*, 180–194. https://doi.org/10.1016/j.jobe.2018.03.005

[110] Li, Y. G., Zhang, M. Y., Li, Y., Li, Q.S., & Liu, S. J. (2020). Experimental study on wind load characteristics of high-rise buildings with opening. *The structural design of tall and special buildings*, *29*(9), e1734. https://doi.org/10.1002/tal.1734

[111] Elshaer, A., & Bitsuamlak, G. (2018). Multiobjective aerodynamic optimization of tall building openings for wind-induced load reduction. *Journal of structural engineering*, *144*(10), 4018198. https://doi.org/10.1061/(ASCE)ST.1943-541X.0002199

[112] Marsland, L., Nguyen, K., Zhang, Y., Huang, Y., Abu-Zidan, Y., Gunawardena, T., & Mendis, P. (2022). Improving aerodynamic performance of tall buildings using façade openings at service floors. *Journal of wind engineering and industrial aerodynamics*, *225*, 104997. https://doi.org/10.1016/j.jweia.2022.104997

[113] Sharma, A., Mittal, H., & Gairola, A. (2019). Detached-eddy simulation of interference between buildings in tandem arrangement. *Journal of building engineering*, *21*, 129–140. https://doi.org/10.1016/j.jobe.2018.10.004

[114] Germi, M. S., & Kalehsar, H. E. (2021). *Numerical investigation of interference effects on the critical wind velocity of tall buildings.* Structures (Vol. 30, pp. 239–252). https://doi.org/10.1016/j.istruc.2021.01.013

[115] Elshaer, A., Gairola, A., Adamek, K., & Bitsuamlak, G. (2017). Variations in wind load on tall buildings due to urban development. *Sustainable cities and society*, *34*, 264–277. https://doi.org/10.1016/j.scs.2017.06.008

[116] Braun, A. L., & Awruch, A. M. (2009). A partitioned model for fluid--structure interaction problems using hexahedral finite elements with one-point quadrature. *International journal for numerical methods in engineering*, *79*(5), 505–549. https://doi.org/10.1002/nme.2566

[117] Thepmongkorn, S., Kwok, K. C. S., & Lakshmanan, N. (1999). A two-degree-of-freedom base hinged aeroelastic (BHA) model for response predictions. *Journal of wind engineering and industrial aerodynamics*, *83*(1), 171–181. https://doi.org/10.1016/S0167-6105(99)00070-7

[118] Zhang, Y., Habashi, W. G., & Khurram, R. A. (2015). Predicting wind-induced vibrations of high-rise buildings using unsteady CFD and modal analysis. *Journal of wind engineering and industrial aerodynamics*, *136*, 165–179. https://doi.org/10.1016/j.jweia.2014.11.008

[119] Hasama, T., Saka, T., Itoh, Y., Kondo, K., Yamamoto, M., Tamura, T., & Yokokawa, M. (2020). Evaluation of aerodynamic instability for building using fluid--structure interaction analysis combined with multi-degree-of-freedom structure model and large-eddy simulation. *Journal of wind engineering and industrial aerodynamics*, *197*, 104052. https://doi.org/10.1016/j.jweia.2019.104052

[120] Feng, C., Fu, G., & Gu, M. (2022). Numerical simulation of wind veering effects on aeroelastic responses of thousand-meter-scale super high-rise buildings. *Journal of building engineering*, *46*, 103790. https://doi.org/10.1016/j.jobe.2021.103790

[121] Marukawa, H., Kato, N., Fujii, K., & Tamura, Y. (1996). Experimental evaluation of aerodynamic damping of tall buildings. *Journal of wind engineering and industrial aerodynamics*, *59*(2–3), 177–190. https://doi.org/10.1016/0167-6105(96)00006-2

[122] Zheng, C., Xie, Y., Khan, M., Wu, Y., & Liu, J. (2018). Wind-induced responses of tall buildings under combined aerodynamic control. *Engineering structures*, *175*, 86–100. https://doi.org/10.1016/j.engstruct.2018.08.031

[123] Péntek, M., Winterstein, A., Vogl, M., Kupás, P., Bletzinger, K. U., & Wüchner, R. (2018). A multiply-partitioned methodology for fully-coupled computational wind-structure interaction simulation considering the inclusion of arbitrary added mass dampers. *Journal of wind engineering and industrial aerodynamics*, *177*, 117–135. https://doi.org/10.1016/j.jweia.2018.03.010

[124] Wijesooriya, K. D. (2021). *An uncoupled fluid-structure interaction numerical framework to estimate wind induced loads on super-tall structures*. [Thesis]. https://B2n.ir/pj5909

[125] Ricci, M., Patruno, L., Kalkman, I., De Miranda, S., & Blocken, B. (2018). Towards LES as a design tool: Wind loads assessment on a high-rise building. *Journal of wind engineering and industrial aerodynamics*, *180*, 1–18. https://doi.org/10.1016/j.jweia.2018.07.009

[126] Jamesahar, E., Sabour, M., Shahabadi, M., Mehryan, S. A. M., & Ghalambaz, M. (2020). Mixed convection heat transfer by nanofluids in a cavity with two oscillating flexible fins: A fluid–structure interaction approach. *Applied mathematical modelling*, *82*, 72–90. https://doi.org/10.1016/j.apm.2019.12.018

[127] Onodera, N., Aoki, T., Shimokawabe, T., & Kobayashi, H. (2013). Large-scale LES wind simulation using lattice Boltzmann method for a 10 km$\times$ 10 km area in metropolitan Tokyo. *Tsubame 9*(2), 1–8. https://doi.org/10.1016/j.jweia.2018.07.009

[128] Jacob, J., & Sagaut, P. (2018). Wind comfort assessment by means of large eddy simulation with lattice Boltzmann method in full scale city area. *Building and environment*, *139*, 110–124. https://doi.org/10.1016/j.buildenv.2018.05.015

[129] King, M. F., Khan, A., Delbosc, N., Gough, H. L., Halios, C., Barlow, J. F., & Noakes, C. J. (2017). Modelling urban airflow and natural ventilation using a GPU-based lattice-Boltzmann method. *Building and environment*, *125*, 273–284. https://doi.org/10.1016/j.buildenv.2017.08.048

[130] Buffa, E., Jacob, J., & Sagaut, P. (2021). Lattice-Boltzmann-based large-eddy simulation of high-rise building aerodynamics with inlet turbulence reconstruction. *Journal of wind engineering and industrial aerodynamics*, *212*, 104560. https://doi.org/10.1016/j.jweia.2021.104560

[131] Han, M., Ooka, R., & Kikumoto, H. (2020). Validation of lattice Boltzmann method-based large-eddy simulation applied to wind flow around single 1: 1: 2 building model. *Journal of wind engineering and industrial aerodynamics*, *206*, 104277. https://doi.org/10.1016/j.jweia.2020.104277

[132] Camps Santasmasas, M., Zhang, X., Parslew, B., Lane-Serff, G. F., Millar, J., & Revell, A. (2022). Comparison of lattice Boltzmann and Navier-Stokes for zonal turbulence simulation of urban wind flows. *Fluids*, *7*(6), 181. https://doi.org/10.3390/fluids7060181

[133] Mays, R. G., Orzech, L. S., Ciarfella, W. A., & Phillips, R. W. (1985). PDM: A requirements methodology for software system enhancements. *IBM systems journal*, *24*(2), 134–149.

[134] Salehi, H., & Burgueño, R. (2018). Emerging artificial intelligence methods in structural engineering. *Engineering structures*, *171*, 170–189. https://doi.org/10.1016/j.engstruct.2018.05.084

[135] Adeniran, A. O. (2016). Impacts of the fourth industrial revolution on transportation in the developing nations. *International educational scientific research journal*, *2*(11), 56–60. https://B2n.ir/fh3357

[136] Adeniran, A. O. (2017). Idea of smart development in the fourth industrial revolution emphasis on smart road. *SciFed Journal of Telecommunication*, *1*, 1–6. https://B2n.ir/qr9810

[137] Martinez-Vázquez, P., & Rodriguez-Cuevas, N. (2007). Wind field reproduction using neural networks and conditional simulation. *Engineering structures*, *29*(7), 1442–1449. http://dx.doi.org/10.1016/j.engstruct.2006.08.024

[138] Wu, T., & Kareem, A. (2011). Modeling hysteretic nonlinear behavior of bridge aerodynamics via cellular automata nested neural network. *Journal of wind engineering and industrial aerodynamics*, *99*(4), 378–388. https://doi.org/10.1016/j.jweia.2010.12.011

[139] Raissi, M., Wang, Z., Triantafyllou, M. S., & Karniadakis, G. E. (2019). Deep learning of vortex-induced vibrations. *Journal of fluid mechanics*, *861*, 119–137. https://doi.org/10.1017/jfm.2018.872

[140] Hu, G., & Kwok, K. C. S. (2020). Predicting wind pressures around circular cylinders using machine learning techniques. *Journal of wind engineering and industrial aerodynamics*, *198*, 104099. https://doi.org/10.1016/j.jweia.2020.104099

[141] Lin, P., Hu, G., Li, C., Li, L., Xiao, Y., Tse, K. T., & Kwok, K. C. S. (2021). Machine learning-based prediction of crosswind vibrations of rectangular cylinders. *Journal of wind engineering and industrial aerodynamics*, *211*, 104549. https://doi.org/10.1016/j.jweia.2021.104549

[142] Kim, B., Yuvaraj, N., Tse, K. T., Lee, D.-E., & Hu, G. (2021). Pressure pattern recognition in buildings using an unsupervised machine-learning algorithm. *Journal of wind engineering and industrial aerodynamics*, *214*, 104629. https://doi.org/10.1016/j.jweia.2021.104629

[143] Lin, P., Ding, F., Hu, G., Li, C., Xiao, Y., Tse, K. T., … & Kareem, A. (2022). Machine learning-enabled estimation of crosswind load effect on tall buildings. *Journal of wind engineering and industrial aerodynamics*, *220*, 104860. https://doi.org/10.1016/j.jweia.2021.104860

[144] Chen, F. B., Wang, X. L., Li, X., Shu, Z. R., & Zhou, K. (2022). Prediction of wind pressures on tall buildings using wavelet neural network. *Journal of building engineering*, *46*, 103674. https://doi.org/10.1016/j.jobe.2021.103674

[145] Hu, G., Liu, L., Tao, D., Song, J., Tse, K. T., & Kwok, K. C. S. (2020). Deep learning-based investigation of wind pressures on tall building under interference effects. *Journal of wind engineering and industrial aerodynamics*, *201*, 104138. https://doi.org/10.1016/j.jweia.2020.104138

[146] Kim, B., Lee, D.-E., Preethaa, K. R. S., Hu, G., Natarajan, Y., & Kwok, K. C. S. (2021). Predicting wind flow around buildings using deep learning. *Journal of wind engineering and industrial aerodynamics*, *219*, 104820. https://doi.org/10.1016/j.jweia.2021.104820

[147] Huang, S. H., Li, Q. S., & Wu, J. R. (2010). A general inflow turbulence generator for large eddy simulation. *Journal of wind engineering and industrial aerodynamics*, *98*(10–11), 600–617. https://doi.org/10.1016/j.jweia.2010.06.002