Applied Mathematics & Information Sciences
An International Journal

http://dx.doi.org/10.18576/amis/180518

Numerical Simulation of Fluid-Structure Interactions on Super-Tall Slender Structures Using Two-Way Coupling Techniques

Amira Mohamed Hussin

Department of Mathematics, College of Science and Humanities, Prince Sattam bin Abdulaziz University, Al-Kharj 16273, Saudi Arabia

Received: 17 Feb. 2024, Revised: 4 Jun. 2024, Accepted: 7 Jun. 2024
Published online: 1 Sep. 2024

Abstract: Super-tall slender structures are increasingly prevalent in modern architecture, but their stability and performance are heavily influenced by fluid-structure interactions (FSI) induced by wind loads. Accurate simulation of these interactions is crucial for ensuring structural integrity and safety. This study aims to conduct numerical simulations of FSI on super-tall slender structures using advanced two-way coupling techniques. The primary objective is to develop a comprehensive understanding of the complex interactions between fluid flow and structural response to inform design and optimization strategies. The novelty of this study lies in the utilization of two-way coupling techniques, which allow for simultaneous and iterative solving of fluid dynamics and structural mechanics equations. By incorporating bidirectional feedback between the fluid and structure, this approach captures intricate FSI phenomena with high fidelity, enhancing the accuracy of predictions. The proposed framework involves integrating computational fluid dynamics (CFD) and structural analysis methods within a cohesive simulation environment. The CFD solver models turbulent airflow around the super-tall structure, while the structural solver accounts for the deformation and response of the structure under fluid loading. The coupling between the two solvers enables mutual influence and interaction. Numerical simulations are performed for a representative super-tall slender structure subjected to varying wind conditions. The simulations capture detailed fluid flow patterns, structural deformations, and dynamic responses. Analysis of the results reveals the intricate coupling between fluid dynamics and structural behavior, highlighting the importance of considering FSI effects in structural design. The study demonstrates the effectiveness of two-way coupling techniques in accurately simulating FSI on super-tall slender structures. The insights gained from the simulations contribute to a deeper understanding of the dynamic behavior of these structures under wind loading. Ultimately, this research facilitates the development of more resilient and efficient design practices for super-tall buildings in complex wind environments.

Keywords: Fluid-Structure Interactions, Super-Tall Structures, Two-Way Coupling, Computational Fluid Dynamics (CFD), Structural Dynamics.

1 Introduction

Super-tall slender buildings, sometimes referred to as super-slim or ultra-slender skyscrapers, are a unique and eye-catching class of high-rise structures that are gaining more and more attention in the fields of urban planning and architecture. These buildings are notable for their extraordinary height in relation to their comparatively small footprints. Their remarkable height-to-width ratios, which can significantly surpass those of conventional skyscrapers, characterise them architecturally. These ratios frequently exceed 10:1 and occasionally even approach 20:1 or higher [1] . The emergence of ultra-tall, thin structures can be attributed to the difficulties urban areas face in accommodating growing populations on constrained land. By providing a solution to the need for vertical expansion in crowded cities, these structures help developers and architects make the most use of the limited amount of available urban space. Building extremely tall, extremely thin structures is a difficult task with many moving parts. Because these buildings are vulnerable to unique challenges like wind-induced sway and structural stresses, creative engineering and design solutions are needed to ensure their stability and safety. Architects and engineers use state-of-the-art technologies that innovative building components, and sophisticated

* Corresponding author e-mail: a.hussin@psau.edu.sa

substances to address these problems [2]. The use of tuned mass dampers, which are counterweights built into the building's design, is one notable aspect of extremely tall and thin buildings. By opposing wind forces and lowering vibrations, these dampers support the preservation of the building's structural integrity and occupants' comfort. The world's tallest building, the Burj Khalifa in Dubai, the Petronas Towers in Kuala Lumpur, and the Shanghai Tower in China are a few of the most famous instances of extremely tall, thin buildings [3] . In addition to changing the skylines of the cities where they stand, these skyscrapers push the envelope of contemporary architectural design and act as symbols of human achievement. Extremely tall, thin buildings are adaptable and can accommodate a wide range of uses, including observation decks, commercial and residential spaces, and more. Their astounding design and technical skill have captured the imagination of people all over the world, not only redefining the urban landscape but also providing breathtaking and iconic landmarks [4] . Both one-way and two-way coupling methods are widely used in many scientific and engineering applications, and they are essential to numerical simulations of fluid-structure interactions (FSI). These methods are used to simulate and comprehend the intricate relationships that exist between a fluid (like water or air) and a structure (like a bridge or building) when they are exposed to dynamic forces (like water or wind flow) [5] . A sequential method where the fluid and structural domains are solved independently is used in one-way coupling. This method takes into account how the fluid affects the structure, but it does not explicitly model the opposite interaction, in which the structure affects the fluid. This method is less computationally challenging and is frequently employed when the fluid's impact on the structure outweighs the structure's influence on the fluid [6]. On the other hand, the governing equations for the fluid and the structure are simultaneously solved in two-way coupling, also referred to as fully coupled simulation. This method takes into consideration the two-way interaction, which implies that the motion of the structure affects the surrounding fluid in addition to the fluid's impact on the structure. Two-way coupling is more computationally demanding, but it offers a more accurate representation of the fluid-structure dynamic feedback [7]. For situations where these interactions are important, like super-tall, thin structures subjected to turbulent winds, two-way coupling is necessary. The particular simulation scenario will determine which one-way or two-way coupling technique is best. Factors such as fluid-structure interaction type, computational capacity, and accuracy will all be taken into account. By comparing and contrasting these methods, researchers and engineers can decide which is best for a particular issue. This process advances research knowledge of how structures respond to dynamic environmental conditions and informs the design and safety protocols for a variety of applications, such as extremely tall and thin urban structures [8]. In software engineering, coupling means how closely different parts of a system are connected, and it can be categorized as either low or high. Low coupling types are when modules pass data between each other or have a common data structure. In medium coupling, there are two kinds of connections between modules: control coupling, where modules communicate by using control flags, and common coupling, where modules share global data [9]. Content coupling is a stronger kind of connection, where modules know about each other's insides (Figure 1). High coupling types include relying on things outside of the code and having an execution order that depends on time. Reducing coupling means creating software that is easier to manage and update. When there is less coupling, it is easier to make changes without causing problems. On the other hand, too much coupling can make software more fragile and difficult to maintain. Coupling techniques in software engineering determine how much modules or parts of a system rely on each other. There are different ways to connect things together, and research can group them based on how strongly they are connected. Low coupling techniques are ways to reduce how much modules in a system depend on each other. Data coupling is when modules pass information to each other without needing to know how the other module is set up. Stamp coupling is when modules share data structures to communicate with each other [10]. Control coupling is when modules use shared control flags to communicate, but there can be one-way coupling if one module affects the other without a response. Medium coupling techniques involve a moderate level of connection between different parts of a system. One example is common coupling, where multiple parts of the system use the same global data without relying on each other. Content coupling occurs when modules are closely dependent on each other's inner structures. High coupling techniques involve two types: external coupling and temporal coupling [2]. External coupling means that modules depend on external entities, such as shared databases. Temporal coupling means that modules need to be executed in a specific order because they have time dependencies. These advanced coupling techniques frequently involve a connection in both directions, as both modules rely on each other or external elements. In simple terms, one-way coupling is usually seen in low coupling techniques like data, stamp, and control coupling. In these techniques, modules interact with very little dependency on each other. Two-way coupling is more typical in medium to high coupling techniques, such as common, content, external, and temporal coupling, where modules might depend on each other a lot. It is better to have lower coupling when creating software because it makes it easier to maintain and modify [3] . The key contribution of the research are mentioned as follows:

– The study introduces a novel numerical framework for analysing wind-induced loads on tall structures,

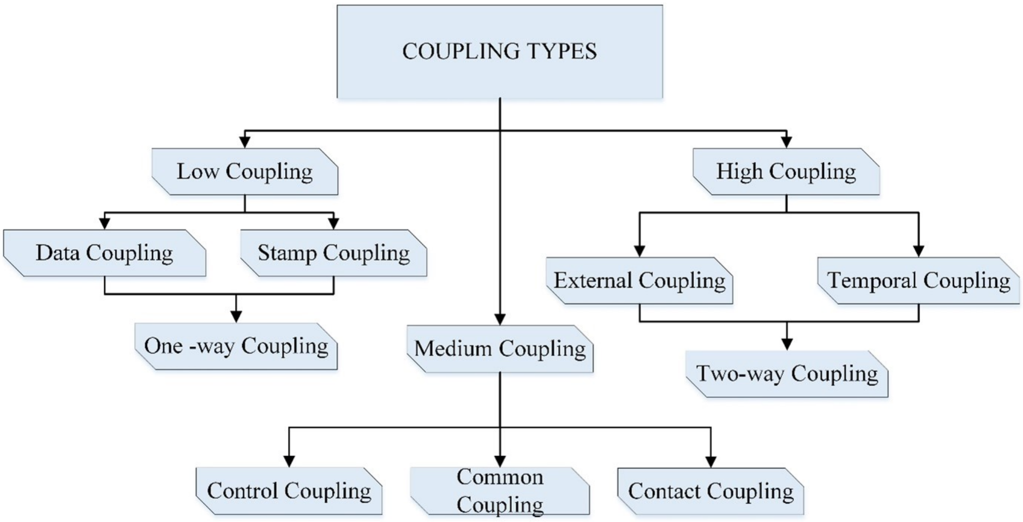

Fig. 1: Coupling Types

offering a digital simulation alternative to elastic wind tunnel testing.

– By employing advanced two-way coupling techniques, the research enables simultaneous modelling of turbulent airflow dynamics and structural responses, enhancing understanding of fluid-structure interactions.
– Validation through experimental testing in a boundary layer wind tunnel confirms the accuracy and reliability of the proposed numerical framework, ensuring its effectiveness in predicting structural responses under varying wind conditions.
– The developed methodology achieves comparable precision to traditional vortex simulation techniques while significantly reducing computational costs, making it a practical and efficient tool for engineering applications.

This work provides comparative analysis of one-way and two-way coupling techniques in numerical simulations, specifically focusing on their application to fluid-structure interactions in super-tall slender structures. In addition, the study systematically evaluates the strengths and weaknesses of each coupling method under various conditions. Moreover, the study introduces novel methodologies for simulating fluid-structure interactions in the context of super-tall slender structures.

2 Related Works

The investigation of wind-induced loads on extraordinarily tall structures has received extensive attention in the domains of structural, wind, and civil engineering. Wind tunnel testing, like aeroelastic modelling, is becoming increasingly necessary as the structural design of these extremely tall structures becomes more complex. However, due to the complexity of the testing process, elastic wind tunnel testing is often overlooked at the project concept stage. Therefore, for industrial use, a different approach in terms of digital simulation is essential. This research provides a numerical framework for estimating the structural response of thin and very high wind-sensitive structures to achieve this goal. The system is based on the Fluid-Structure Interaction (FSI) strategy, which gauges auxiliary reactions by performing a temporal auxiliary evaluation and employments computational liquid elements (CFD) to anticipate wind-induced loads on the building's system. The information system displayed in this proposal is supplemented by three exploratory tests performed in a boundary layer wind burrow (BLWT) for approval purposes. The proposed numerical CFD procedure employments coordinates whirlpool recreation (ELES) and illustrates that the strategy can accomplish comparative precision at a division of the taken a toll required for expansive vortex reenactment (LES) investigation. The center of this system proposes a novel and profoundly proficient weight mapping strategy, which presents a decoupled one-dimensional FSI reenactment. The numerical system is fundamentally assessed based on existing numerical strategies and multi-degree-of-freedom (MDOF) aeroelastic wind burrow testing. It is illustrated that the developed strategy is able of anticipating comparable basic reactions in a time-efficient way. In outline, this proposition gives a comprehensive numerical system that architects can effortlessly apply to decide the basic reaction of wind-sensitive structures [10].

Utilizing Deferred Decoupled Vortex Recreation (DDES) based on the SpalartAllmaras (SA) turbulence show, liquid structure interaction (FSI) reenactments are performed for the 1-1/2 high-speed compressor deck GE. The insecure compressed Navier-Stokes conditions are spatially sifted with the SA-DDES turbulence demonstrate and the auxiliary conditions of movement are unravelled in a completely coupled way. For invariant stream, the low-diffusion Riemann E-CUSP solver with preservationist third-order WENO remaking factors is utilized; For gooey streams, a second-order central separation plot is utilized. The excitation constrain is created at a non-driving recurrence due to the precariousness of the vortex tip. There was fabulous assertion between stage exploratory estimations and anticipated NSV frequencies. Patel et al. [11] research is centred on examining the asynchronous vibration in stage 1-1/2 of a high-speed axial compressor through the use of a fully coupled fluid-structure interaction technique. When conducting the recreation, the 1/7 ring area and the DDES turbulence show are actualized, alongside completely preservationist sliding boundary conditions between the edge lines. The incorporation of time-shifted stage interpretation conditions at the circumferential boundaries facilitates computational efforts. Operational soundness at the boundary is kept up, minimizing slows down due to precariousness of the moving head vortex. Eminently, a tornado-shaped vortex shows up close the driving edge of the rotor, crossing the suction surface of one edge some time recently moving toward the adjoining edge within the inverse heading of turn. This vortex-induced movement leads to unstable edge stacking and causes engine-irrelevant vibrations, particularly close the primary torsional mode. Within the edge tip locale, irregular vortex (RV) event is watched occasionally. The anticipated non-synchronous vibration (NSV) recurrence of 2,595 Hz concurs well with the measured recurrence of 2,600 Hz amid seat testing. In spite of the promising agreement between anticipated and measured frequencies, decreasing computational exertion through time-shifted stage interpretation conditions may result in a few misfortunes of exactness within the calculation. get better points of interest or subtleties of fluid–structure intelligent.

Auxiliary wellbeing checking (SHM) frameworks check for harm without causing hurt to the structure. The development of ultrasonic waves is followed and analyzed as portion of auxiliary wellbeing observing to distinguish any changes within the wave design. These changes help figure out where a building is damaged, if at all. De et al. [12] improve the current research by considering the possibility of not being sure about the materials and shape of a building. This considers employments a material science show that combines flexible and acoustic wave conditions. It's called the wave engendering in fluid–structure and their interface (WpFSI) issue. Since it takes a parcel of computer control to reenact the WpFSI issue numerous times, investigate made distant better; a higher; a stronger; an improved"¿a distant better way to anticipate how UGW moves in a liquid and a structure when we're not beyond any doubt what will happen. Investigate used machine learning methods to do this. To begin with, inquire about make a number of illustration pictures employing a complicated computer program that appear what the questionable parts of the structure might see like. To begin with, investigate utilize Gaussian forms to prepare with these pictures. At that point investigate utilize convolutional neural systems to foresee the wave and make the pictures see way better. This makes a difference us to make high-quality pictures of the wave designs when there are vulnerabilities. The discoveries appear that the recommended strategy can make a precise figure for the WpFSI issue indeed when there's instability. In the work of Lakshmynarayanana and Hirdaris [8], Through two-way coupling and evaluation in flow, computational modelling based on the fluid-structure interaction (FSI) method is used to study the hydrodynamic properties of the bottom trawl system. channel range determined by modifying Tauti's law. The $k-\omega$ turbulent shear push (SST) demonstration was used to depict the stream in this numerical show, while the limited volume approach was employed to understand the Navier-Stokes conditions. The limited component strategy was utilized to illuminate the expansive strain nonlinear basic energetic conditions to portray the trawl arrangement and nodal relocations. Calculate the channel zone employing a adjusted form of Tauti's law. To further approximate the effect of turbulence on foot trawl execution, three-dimensional (3D) acoustic Doppler anemometer (ADV) estimations were carried out.

A strong agreement between the test data and the numerical results is revealed by the comparison. Both the use of nylon monofilament to reduce the utilised breadth and Dyneema multifilament to expand the work determine result in a decrease in malleable drive of approximately 2. 1 times, individually, and 22 times (p ¡ 0. 002 ANCOVA test), according to numerical and exploratory results. It is discovered that the turbulent boundary layer stream is spatially advancing around the trawl and that the vortex is visible behind the trawl. Notoriety vortex-induced vibrations (VIV) will result from vortex current misfortune caused by stream partition at the windward corners of a high-rise building. In order to consider the wind-induced responses of structures, this paper suggests an effective and underutilised two-dimensional fluid-structure interaction (FSI) strategy known as the comparable mass structure (ELMS) strategy. House of the Joint Flying Counselling Chamber (CAARC). The numerical comes about of ELMS are approved against accessible aeroelastic testing information. In arrange to affirm the computational effectiveness of the ELMS strategy, this think about too consolidates an appraisal of two other two-dimensional coupled FSI strategies (the FFD strategy and a mapping addition calculation). (MIASC)) to recreate the reaction of the same high-rise building. Besides, the Vibrational Initiated Vortex (VIV) component in high-rise structures is completely inspected based on numerical discoveries gotten through the Work of Expansive Vortex Reenactment (ELMS) strategy. The results demonstrate that the ELMS strategy adeptly captures the articulated intensification of cross-wind reactions and the event of the ”locking” wonder inside the basic wind speed extend. Strikingly, the computational proficiency of the ELMS strategy outperforms that of the other two Fluid-Structure Interaction (FSI) strategies. Amid the locking organize, there’s a notable improvement within the spatial relationship and ghostly coherence of neighbourhood loads at different statures of the building. These discoveries contribute altogether to a all-encompassing comprehension of the VIV marvel in high-rise structures and offer an proficient two-way coupled FSI method for engineers and analysts locked in within the versatile plan against wind for slim high-rise buildings [13]. All of these studies are focused on examining how different structures react to different engineering situations and the possible harm they might suffer. Studying how wind affects tall buildings, how trawls work underwater, and how vibrations impact high-rise buildings all deal with how fluids and structures interact with each other, and it’s all very complicated. Numerical simulations are a good option instead of complex and costly experiments. But the challenge is to find the right balance between speed and accuracy in the computer calculations. These studies show new ways of doing things, like using a simulation called Embedded-Large Eddy Simulation (ELES) and making a computer program that uses machine learning. Despite these improvements, it’s still difficult to balance being accurate and not using too much computer power. It’s important to make reliable predictions even when things are uncertain and there are complicated physical interactions.

3 Problem Statement

The studies highlighted in the provided text shed light on the intricate interactions between fluids and structures in various engineering scenarios, including wind effects on tall buildings, underwater trawl dynamics, and vibrations in high-rise structures. Despite the advancements in numerical simulation techniques, there exists a critical research gap in achieving a balance between computational efficiency and accuracy when predicting fluid-structure interactions (FSI) under uncertain conditions. The existing research primarily focuses on utilizing numerical simulations as an alternative to complex and costly experiments, leveraging techniques such as Embedded-Large Eddy Simulation (ELES) and machine learning algorithms to enhance computational efficiency and prediction accuracy. However, achieving reliable predictions while considering uncertainties and complex physical interactions remains a significant challenge. The scope of future research lies in addressing this research gap by developing advanced numerical simulation methods that strike a balance between computational efficiency and accuracy in predicting FSI phenomena. This includes further refinement of simulation techniques such as ELES and machine learning-based approaches to improve their predictive capabilities under uncertain conditions. Additionally, exploring innovative strategies for optimizing computational resources while maintaining simulation fidelity will be crucial. Ultimately, bridging this gap will enable engineers and researchers to make more reliable predictions of fluid-structure interactions in diverse engineering applications, leading to enhanced design and performance optimization of complex engineering systems.

4 Methodology

In the past, it was hard to use computer models to study how liquids and structures interact in really tall buildings. One big challenge was figuring out which technique to use: one-way or two-way coupling. The complex structures and their sensitivity to wind forces means research need to carefully look at how they are connected together. One-way coupling is when fluid and structure simulations are separated and updated one after the other. It’s often used because it’s faster to calculate. But, this way of doing things often ignores how the fluid and structure affect each other, which could make the results less accurate. This study wants to compare how well different ways of connecting fluids and structures work in computer simulations of tall, thin buildings. This will help us understand how to solve the problems research have with these kinds of buildings. Research want to do this research because it’s really important to find a better way to predict how these complex systems will respond. The scope of the study includes an in-depth analysis of the advantages and disadvantages of the two docking techniques, taking into account factors such as computational cost, accuracy, and real-world applicability. The novelty of the proposed method lies in the comprehensive comparison of one-dimensional and two-dimensional grafting techniques, with a special focus on highly slender structures. By combining advanced numerical methods and drawing on the latest developments in fluid dynamics and structural analysis, this study aims to provide an in-depth understanding of the trade-offs involved. This study makes a valuable contribution to the field by providing engineers and researchers with insight into the most effective coupling techniques for predicting fluid–structure interactions in thin structures. very high, thereby enhancing the cutting edge in this important research field.

4.1 Fluid Dynamics

The Navier-Stokes equations form the cornerstone of fluid dynamics, delineating the fundamental principles governing the motion of viscous fluids. In their three-dimensional manifestation, these partial differential equations articulate the conservation of mass and the momentum balance for each velocity component in a fluid. The continuity equation ensures mass conservation, while the momentum equations express the intricate interplay between pressure, inertia, and viscosity. These equations capture the evolution of fluid velocity over time, providing a comprehensive framework to understand the complex dynamics of fluid flow. Notably, the Navier-Stokes equations are vital for simulating a diverse array of phenomena, from the laminar flow in pipelines to the turbulent currents around aerodynamic surfaces, playing a pivotal role in unravelling the intricate behaviors of fluids in motion.

4.1.1 Navier-Stokes Equation

Over time, sub grid-scale (SGS) models for large-scale fault simulation (LES) have evolved in three different forms: algebraic eddy viscosity models, one-equation eddy viscosity models, and two-equation eddy viscosity models. The main objective of this development is to accurately represent the SGS behavior and energy transfer processes on a grid scale for flows characterized by high Reynolds numbers. Consequently, there is an urgent need to develop them SGS models provide engineering applications that can efficiently solve practical engineering challenges. In line with this objective, the authors present a new single-equation dynamic SGS model for LES ; based on the single-equation dynamic SGS model proposed and the WALE model developed [14]. A brief schematic of this model is provided below: As observed in [15], Newton's second law, F = ma, which states that force is the product of an object's mass times its acceleration, can be applied to the Navier-Stokes equation. Research use shear stress and density in this equation. Density is the measure of the amount of matter in a given volume, and shear stress is the force acting on a material in the same direction as its surface. Suppose that

$$\rho\left[\frac{dx}{dt}+x\cdot\nabla x\right]=\nabla\cdot\sigma+f \tag{1}$$

In Eqn (1) the density of the fluid and is equivalent to mass is denoted as ρ, the acceleration is denoted as $\frac{dx}{dt}+x.\nabla x$ and the velocity is referred as x and the total force is denoted as $\nabla.\sigma+f$ while the shear stress is denoted as $\nabla.\sigma$ and the other force is denoted as f and again the Eqn (1) can be rewritten as:

$$\rho\left[\frac{dx}{dt}+x.\nabla x\right]=-\nabla p+\beta\nabla^2 x+f \tag{2}$$

In Eqn (2) the pressure and dynamic viscosity is denoted as the ρ and μ.

$$\frac{dx}{dt}=-(x.\nabla)\cdot x-\frac{1}{\rho}\nabla p+\beta\nabla^2 x+f \tag{3}$$

Viscosity is a way to quantify how much a fluid resists changing shape under shear stress. By dividing ρ and subtracting $\mathrm{x}\cdot\nabla\mathrm{x}$, from Eqn (3) research can derive the classic form of the Navier-Stokes.

A good discretization in both space and time is necessary for solving this equation. In technical matters, small scale is not in the scope of concern. To reduce computational effort, the solution variables are divided into mean and range values, as in Eqn (4):

$$\omega=\bar{\omega}+\omega' \tag{4}$$

By using Reynolds averaging, research get the Unsteady-Reynolds-Averaged-Navier-Stokes equation which is mentioned in the following Eqn (5):

$$\rho\frac{\partial\bar{x}_m}{\partial t}+\rho\left(\frac{\partial\bar{x}_m\bar{x}_n}{\partial a_n}\right)=\frac{\partial\bar{p}}{\partial a_m}+\eta\frac{\partial}{\partial a_n}\left(\frac{\partial\bar{x}_m}{\partial a_n}+\frac{\partial\bar{x}_n}{\partial a_m}-\frac{2}{3}\frac{\partial\bar{x}_k}{\partial a_k}\delta_{mn}\right)-\rho\left(\frac{\partial}{\partial a_n}\overline{x'_m x'_n}\right) \tag{5}$$

In the above Eqn (5) $x'_m x'_n$ denotes the stress tensor. Research uses a technique known as finite volume to solve the Navier-Stokes and continuity equations. The continuity equation in integral form, Equation (5), is where research begin. The problem consists of three parts: part one depicts the changes in mass within the control volume, part two deals with the changes in mass entering and leaving the control volume, and part three displays the changes in volume. Monitoring the volume inside the form as it transforms [16].

4.2 Two-way coupling

Due to the close integration of the fluid and structural domains in two-way coupling for fluid structure interaction (FSI) simulation, information can be exchanged simultaneously between the two solvers. Unlike one-way joints, two-way joints recognize the mutual influence of the fluid on the structure and vice versa, providing a more accurate representation of the complex interactions and the following diagram (Figure 2) shows the joint two-way connection. For example, when simulating a building's response to seismic forces, the fluid solver (CFD) and structural solver (FEA) interact iteratively at each time step, while updating the fluid flow and structural deformation. This approach is suitable for situations where the interactions among the fluid and the structure have a significant influence on the system's overall efficiency because it considers the interactive response among the two. Although two-way coupling requires more computing power than one-way coupling, it is essential for accurately simulating fluid-structure interactions. It makes the dynamic response of structures to wind transferring and vortex-induced vibration more realistically represented. High-fidelity simulation of fluid-structure interactions requires two-way coupling, even though it requires more computational resources than one-way coupling. This method provides a more accurate depiction of phenomena such as vortex-induced vibration and the dynamic response of structures to wind loading. For a more realistic representation of phenomena like vortex-induced vibration and the dynamic response of structures to wind loading, two-way coupling is crucial in high-fidelity simulation of fluid-structure interactions. Usually, the two-way coupling solution is known for its improved accuracy, especially in situations where the structure bends a lot and the fluid is affected by the bending. Simple two-way connection solutions can make things happen faster and be more stable [17]. Interestingly, energy conservation at the interface is guaranteed by two-way coupling, a feature that one-way coupling does not guarantee. However, one-way coupling simulation has some noteworthy benefits, such as a large computational time savings. Additionally, in the case of one-dimensional coupling, there is no need to explicitly calculate the movement of the fluid mesh, resulting in a mesh that remains of constant quality throughout the simulation. When deciding between one-way and two-way coupling techniques, it is crucial to take these factors into account because the best strategy will rely on the particulars and demands of the substance–structure connection issue. It's resolving liquid.

4.3 Evaluation of Tall Buildings

The assessment of high-rise buildings entails a multidimensional analysis encompassing structural, environmental, economic, and social factors. In the structural realm, these buildings undergo thorough evaluations ensuring stability, load-bearing capacity, and resilience to dynamic forces like wind and seismic activity. Environmental considerations delve into energy efficiency, sustainability, and the building's impact on the surrounding ecosystem. From an economic standpoint, the assessment encompasses the profitability of construction, maintenance, and potential revenue generation. Furthermore, the social impact evaluation examines how high-rise buildings contribute to urban development, public safety, and the general well-being of occupants. Effectively evaluating tall buildings demands a holistic approach that integrates disciplines such as engineering, environmental science, economics, and urban planning. This comprehensive strategy is essential for addressing the unique challenges and opportunities associated with urban development in a vertical context [18].

4.3.1 Design Framework

A structure is subjected to wind loads determined by a combination of effects including local climate, topographic topology, aerodynamic properties and dynamic properties of the structure. In addition, design criteria must be imposed, which take into account the importance of the structure and the level of performance expected based on its use. This sequence of thinking is essential for calculating all the elements needed to create a structure that can endure wind loads, and it is commonly known as "Davenports Wind Load Series" in literature. The diagram illustrates the reproduced Davenports wind load sequence as shown in Figure 3 [19].
The way research think about the force of the wind is very important when designing buildings that could be affected by the wind. Research use weather data to figure out how fast the winds are on average and how fast they can get. This helps us to make rules and services for building things in that area. Final state of stress or load that a structure or material can withstand without failing. Studying how the wind is affected by the land comes next. Research know that the wind on the Earth's surface is affected by how bumpy the land is, which creates the atmospheric boundary layer. The atmospheric boundary layer goes up several hundred meters into the air, and close to the ground the wind speed is slowed down to zero because of friction. The wind gets faster as you go higher up, and eventually it reaches a constant speed where the friction doesn't really affect it. The height where roughness doesn't slow down the wind is called slope altitude (yg). On the other hand, when the wind is close to the ground, it becomes more turbulent because of the friction. But as the wind goes higher, the turbulence decreases. The height of the ABL is usually determined by how bumpy the ground is where the wind blows [20].

4.3.2 Experimental Setup and design

The tower in Figure 4 is based on the KL Tower in Malaysia. It has four main parts: the base, shaft, bulb, and spire. The tower is 406 meters tall and is made up of round sections with different sizes. The tower is 50 meters wide at the base, 15 meters wide in the middle, and 24 meters wide at the top. The tower is 300 meters tall without the spire, and the spire makes up about 1/4 of the total height. The tower is believed to be in an area with medium levels of natural forces.

5 Numerical Simulation

Similar to the test, the computer model used a small size and assumed the building was stiff. Research picked a rectangle for the space and figured out how big it should be by looking at different choices. The area is 13 times longer from front to back, 6 times longer from side to

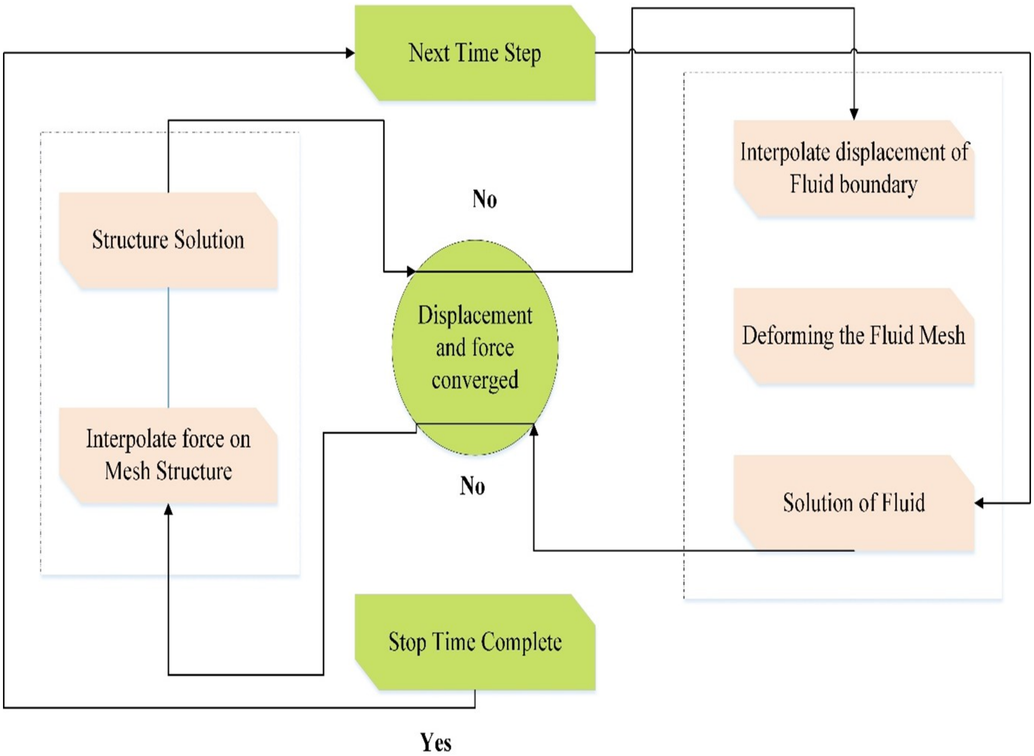

Fig. 2: Two-way Coupling

side, and 3 times longer from bottom to top compared to the height of the building. These boundaries are not very big compared to what Franke and others found. The area's size was mostly determined by how far away the building was from the outlet. In order to reduce edge disruption, it is also advised that the front area to test section ratio should be less than 5%. This particular setup's blockage percentage was less than 1%, which is less than the advised amount. Thus, the method research used to measure the space in this study complies with the specifications to prevent needless flow bending [15]. The airflow around a model may be impacted by the space dividers, which could lead to inaccuracies in the data research gather. The top and both side edges have an impact on how the building flows around it. Modifying the streamlines' path also modifies the vortex's attachment and growth on the windward side. This is when air doesn't flow freely like it usually does. The condition in CFD where no sideways force is applied to the boundary makes the air flow behave like it does in free flow. However, even with these systems, it is important to create boundaries at certain distances to stop the air around the building from moving too fast [21].

5.1 Inlet and Boundary Conditions

Accurately determining input constraints and conditions is critical for obtaining meaningful and realistic results in fluid dynamics simulation. The main factors to consider for input and boundary conditions are what research need to discuss.:

5.1.1 Inlet Conditions

- Velocity Profile: Define the velocity profile of the incoming fluid at the inlet. This may involve specifying a uniform velocity, parabolic velocity distribution, or more complex profiles based on the expected flow characteristics.
- Turbulence Properties: If the flow is turbulent, provide information about the turbulence properties at the inlet, including turbulence intensity and length scale. This is particularly important in simulations where turbulent flow plays a significant role.
- Temperature and Concentration: For simulations involving heat transfer or species transport, specify the temperature and concentration profiles at the inlet.
- Inlet Shape: Consider the shape and geometry of the inlet, ensuring that it accurately represents the physical conditions. In some cases, a contraction or

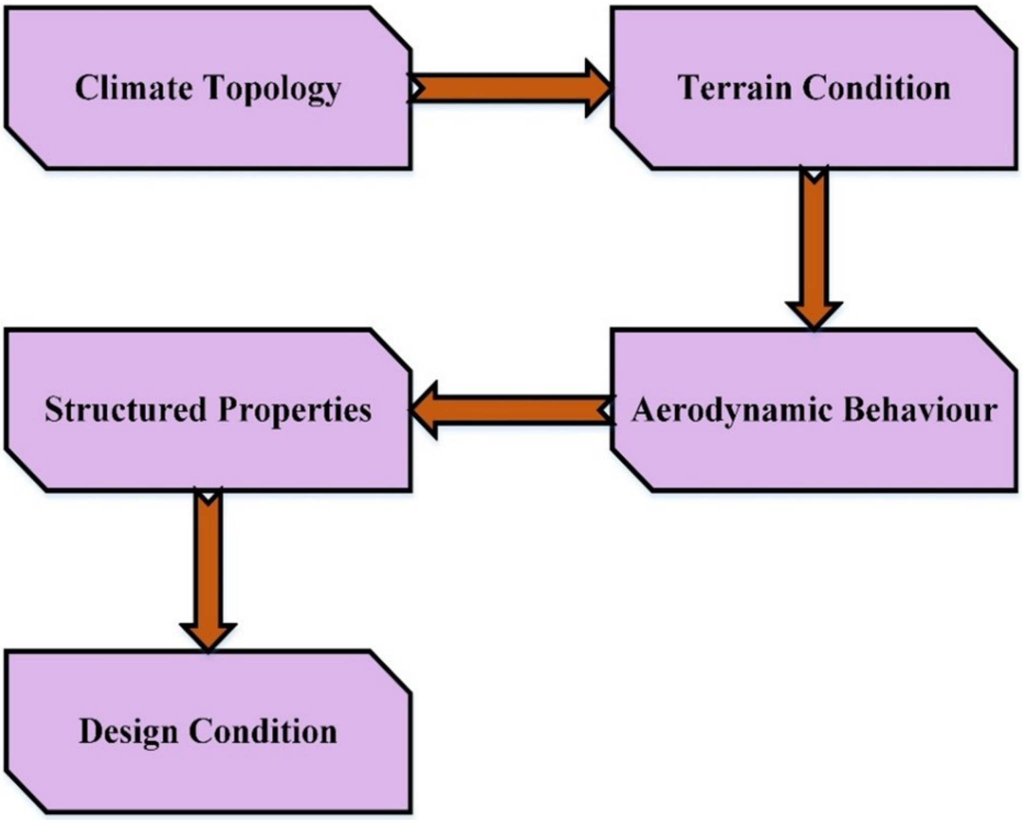

Fig. 3: Wind Loading Chain

diffuser section may be required to establish a fully developed flow.

5.1.2 Boundary Conditions

– No-Slip Condition: For solid boundaries, apply the no-slip condition, meaning that the fluid velocity at the boundary matches the velocity of the boundary itself.
– Pressure Boundary: Specify the pressure at outlets or openings to account for the flow leaving the simulation domain. This is crucial for maintaining mass balance.
– Symmetry: If the geometry exhibits symmetry, use symmetry boundary conditions to reduce computational resources. This is especially useful when modelling only a portion of a symmetric system.
– Free-Slip Condition: In cases where there is slip between the fluid and the boundary, use a free-slip condition, allowing for tangential motion between the fluid and the boundary.
– Far-Field Boundary: In simulations with open domains, set up a far-field boundary condition to represent the conditions far away from the region of interest. This helps to avoid the influence of distant boundaries on the simulated flow.

5.1.3 Special Considerations

– Moving Boundaries: If there are moving boundaries, such as oscillating structures, use dynamic mesh techniques or other methods to account for the changing geometry during the simulation.
– Mass Flow Rate or Velocity Profiles: For simulations where mass flow rates are known, specify them at relevant outlets. Alternatively, prescribe velocity profiles at specific boundaries.
– Adiabatic or Isothermal Conditions: Specify whether the boundaries are adiabatic (no heat transfer) or isothermal (constant temperature).

5.2 Mesh Deformation

Deformation of the liquid-solid boundary results in related deformations in the fluid mesh when two-dimensional coupling computations are used. The displacement diffusion algorithm is a commonly used

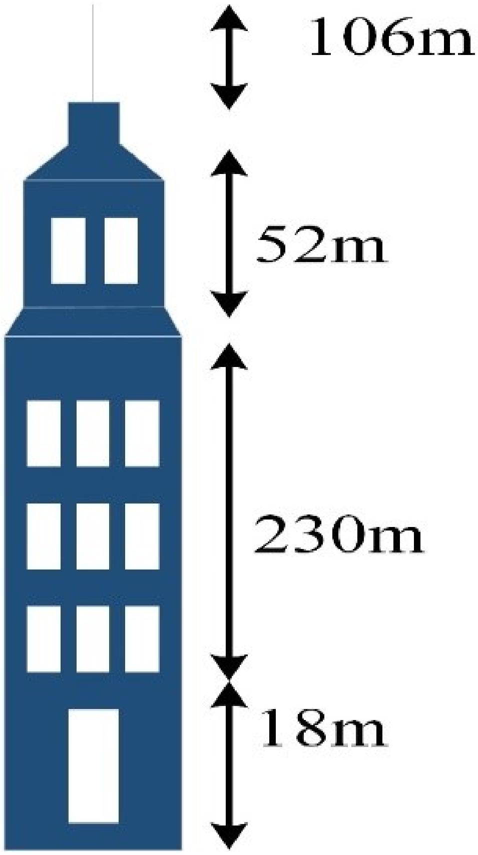

Fig. 4: Tall Building

technique for precisely computing this deformation. The relationships between nodes are represented in this algorithm as springs, and each spring has a unique stiffness value. Interestingly, these springs' stiffness can differ from node to node, but it should be very high in the vicinity of the limits. Strong mesh handling close to fluid structural boundaries is ensured by this high stiffness, which also prevents excessive deformation and preserves mesh integrity [22].
More specifically, the first ten nodes receive removals within the range of the boundary displacement in the normal direction from the fluid-structure boundary. The purpose of this intentional configuration is to maintain a high mesh quality, especially in the boundary layer. The algorithm successfully maintains the structural integrity of the mesh by giving the initial nodes the proper displacements, avoiding distortions that could jeopardise the simulation results' accuracy. The two-way coupling simulations are more stable and reliable overall when a displacement diffusion algorithm with node-specific stiffness values is used, particularly in situations where accurate fluid-structure interaction representation is essential [22].

Selecting appropriate numerical solvers for the fluid and structural domains is crucial for accurately simulating fluid-structure interactions in the context of super-tall slender structures. Here's an explanation of the considerations and potential choices for each domain:
As mentioned before, FSI can involve complex coordination and interface development of CFD and CSD, because different methods are used for CFD and CSD. Furthermore, CFD and CSD are often changed into different forms of code and packages, which makes it difficult to do two-way coupling. In the display test, a method for fluids and structures to interact together is developed using parallel engineering. In Figure 3, the coupling preparation is working as follows:
In order to perform CFD calculations, research ascertain the wind's force
To add up the wind force at different heights, research use a method called integrating the wind loads layer by layer. F_h in Eqn (**??**) is represented as ("Force vector at layer h") and M_h ("torsional moment vector at layer h") are

found for each layer.

$$F_h = \int_h^{h+dh} P\alpha \vec{X} \tag{6}$$

$$M_h = \int_h^{h+dh} \vec{e} \times Pd\vec{X} \tag{7}$$

A message is written using certain codes F_h and M_h, then the message is sent to CSD using socket communication.

After obtaining the values for F_h and M_h, the CSD solver begins calculating CSD to acquire response information such as displacement.

To get the response information at each height level, research integrate the " I_h "displacement and " ω_h " rotation vectors of the out wall layer by layer as mentioned in Eqn (8) to (10):

$$I_h = \frac{1}{n}\sum_{u=1}^{n} \overrightarrow{i_u} \quad u \in (h, h+dh) \tag{8}$$

$$X_u = \frac{1}{n} i_u - I_h \quad u \in (h, h+dh) \tag{9}$$

$$S_u \cdot S_u = r_u \times w_h \cdot S_u \quad u \in (h, h+dh) \tag{10}$$

A message is written using w_h and r_u, then it is sent back to a computer program using a type of communication.

Changing the surface structure of a building in a computer program to improve its performance X_u and w_h, and then doing it again.

The proposed method differs from the conventional two-way coupling approach by exchanging only integrated information of the fluid-structure interface (FSI), rather than exchanging all datasets node by node between computational fluid dynamics (CFD) and computational structural dynamics (CSD). This optimization significantly enhances FSI efficiency. Specifically, for tall buildings, wind-induced deformations predominantly occur globally along the height rather than locally within each layer, justifying the integration of FSI interface information layer by layer. Additionally, employing a socket message transferring method enables the FSI method to couple effectively on heterogeneous platforms, enhancing its versatility and applicability.

6 Result and Discussion

To verify the proposed Fluid-Structure Interaction (FSI) method, a numerical simulation is conducted to analyze wind flow around a 1:375 CAARC building model. The Reynolds number, based on the approaching mean speed UH and model width W, is approximately 70,000. Key boundary conditions and computational parameters for both Computational Fluid Dynamics (CFD) and Computational Structural Dynamics (CSD) are outlined in Table 1. Notably, the CFD and CSD solvers operate on different platforms, leveraging socket communication for efficient data exchange. Additionally, distinct mesh styles are employed by each solver, facilitating the effectiveness of the FSI method. Unlike conventional aeroelastic experiments, the CSD constraint in this simulation is set to a fully elastic state, with the aim of assessing mesh deformation consistency between CFD and CSD. It's important to highlight that a low Young's modulus is utilized for the structure in CSD to simulate significant deformation, allowing rigorous testing of the dynamic mesh method. Fig. 7(a) and (b) illustrates velocity contours with mesh movement from both CFD and CSD, alongside monitored displacements at the top of the building. Fig. 6 depicts force history and spectral density in the FSI simulation, compared with results from a case without FSI.

The mesh nodes in CFD align and deform accurately with beam189 elements on the building's vertical centerline, indicating proper exchange of load and response information in the FSI method. Monitored displacements at the top of the building reveal significant initial along-wind structural response, gradually damping to smaller amplitudes, likely due to aerodynamic and structural damping effects. Conversely, across-wind structural response exhibits continuous amplitude growth until 3s, likely due to synchronization of structural and vortex shedding frequencies. The reduced velocity in the FSI case closely matches the lock-in velocity observed in previous experimental studies. These findings validate the effectiveness and accuracy of the proposed FSI method in capturing complex fluid-structure interactions.

Figure 6 provided depicts the wind displacement mode over time, showcasing the dynamic response of a structure to wind forces. Initially, there's a notable oscillation as the wind exerts its force on the object. However, this oscillation gradually diminishes over time, indicating a damping effect that stabilizes the system. Such insights are invaluable for engineers designing structures to withstand wind loads effectively, ensuring minimal movement and optimal structural integrity.

Figure 7 provided illustrates the wind displacement of a model top over time, revealing the oscillatory response of the system to wind forces. Initially, there is a significant oscillation induced by the wind, followed by a gradual decrease over time, indicating a damping effect that stabilizes the system. Such insights are pivotal for engineers designing structures to endure wind loads effectively and mitigate excessive movement, ensuring optimal structural integrity.

The first instance of testing has a straightforward geometry and a straightforward flow model. The cylindrical regards case has a thin plate fastened to its rear. The ripple current created behind the cylinder influences the plate's behaviour. The complexity of the flow regime can be changed by varying the inlet velocity.

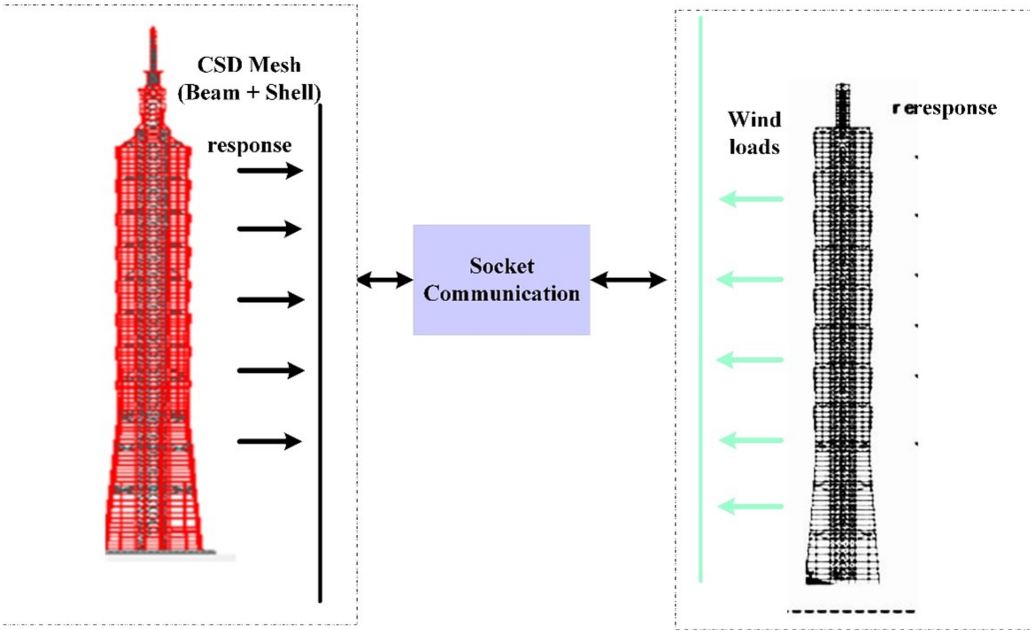

Fig. 5: Two-way FSI

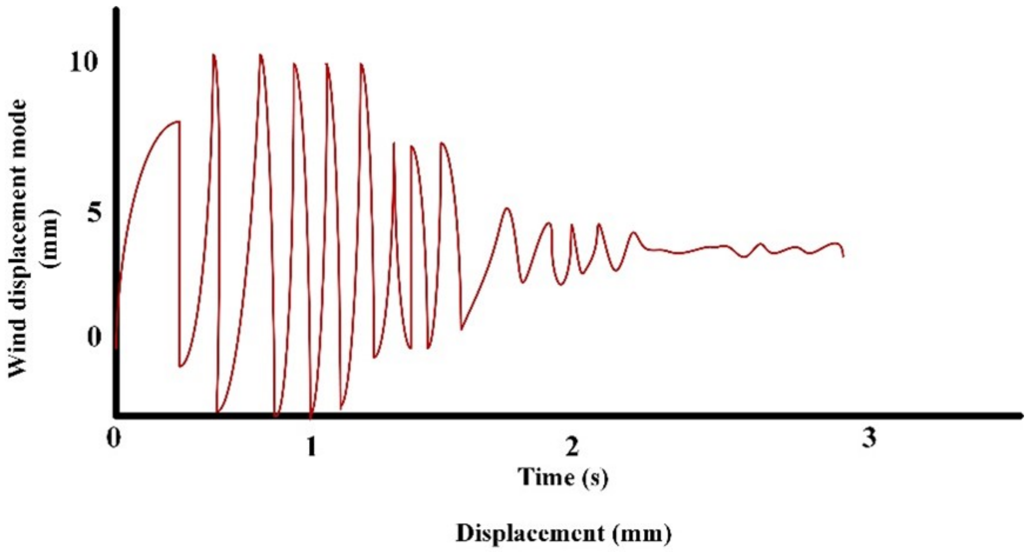

Fig. 6: Wind displacement Mode

Particularly, the surface of the plate is made of a pair of distinct substances, which contributes to the creation of unique natural frequencies. In a square cylinder with a given inlet velocity and a peeling frequency selected either or more distant than the plate's natural frequency, Van Karman vortices form. Figures 6 and 7 illustrate the geometry and its corresponding mesh. Table 1 provides a detailed analysis of different inlet velocities while Table 2 outlines the natural frequencies of the plate. To evaluate the results, the displacement of the back edge of the plate divided by the height of the square cylinder is calculated over time. The analysis also involves applying the fast Fourier transform (FFT) to determine the frequency of the oscillations. It should be noted that unfortunately, no empirical studies have been conducted on this specific issue. Therefore, comparative analysis is performed by

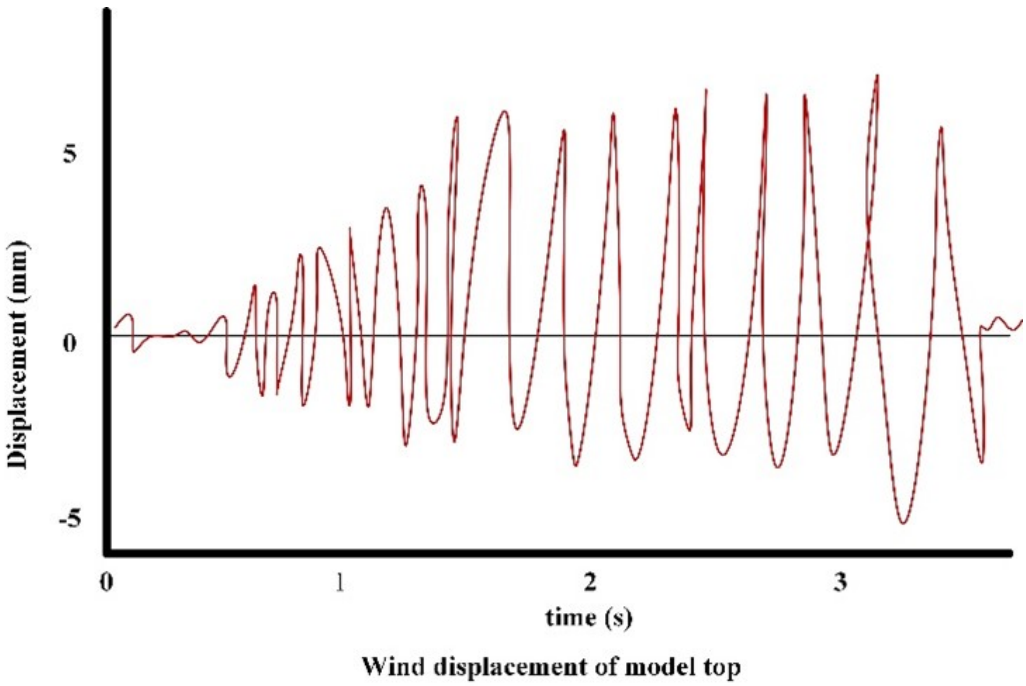

Fig. 7: Wind displacement of Model top

Table 1: Inlet Condition and Parameter

Velocity	Shedding Frequency	Plate
0.28	3.03	Plate 2
0.35	3.69	Plate 1
0.35	3.69	Plate 2
0.65	7.04	Plate 2

Table 2: Natural Frequency among Plate 1 and 2

Natural Frequency (Hz)	
Plate 1	Plate 2
3.08	0.79
15.14	3.04
19.3	3.85
-	10.75
-	18.55

referring to the numerical results, which provide valuable information about the accuracy and reliability of the numerical simulation for this particular test case.

In Case Number 1, Plate 2 is assigned specific material variables and the inlet velocity (v) is 0.28 m/s. The fluid dynamics within the square cylinder with the thin plate attached are significantly influenced by the chosen inlet velocity in this computerised scenario. The relationship between the fluid and the structure becomes more complex due to the unique natural frequencies that are introduced into the system by Plate 2's material properties. Under these specific conditions, the numerical simulation seeks to represent the system's behaviour over time, with a particular emphasis on the displacement of the plate's trailing edge allocated by the square cylinder's height. This displacement over time is computed as part of the evaluation, and the oscillation frequencies are extracted using a fast Fourier transform (FFT). The knowledge gained from this particular case adds to a thorough comprehension of the ways that different material characteristics and inlet velocities affect the dynamics of the fluid-structure communication. It also provides useful information that can be validated and compared against other numerical findings or computational models (see Figures 8).

Figure 9 Two-way Coupling displays a peak around 4 Hz, indicating a significant frequency in the system. This peak suggests the possibility of resonance or optimal coupling, where the system efficiently responds to external forces. Resonant frequencies are crucial in various fields such as structural engineering and mechanical systems. However, a more precise analysis would require additional context or details.

In Figure 10, the graph illustrates the absolute deviation from experimental values for five distinct mesh schemes, showcasing the variations in windward, leeward, crosswind, and average directions. The comparison aims to evaluate the accuracy of the numerical simulations against experimental data. Notably, the transition from Mesh 1 (M1) to Mesh 2 (M2)

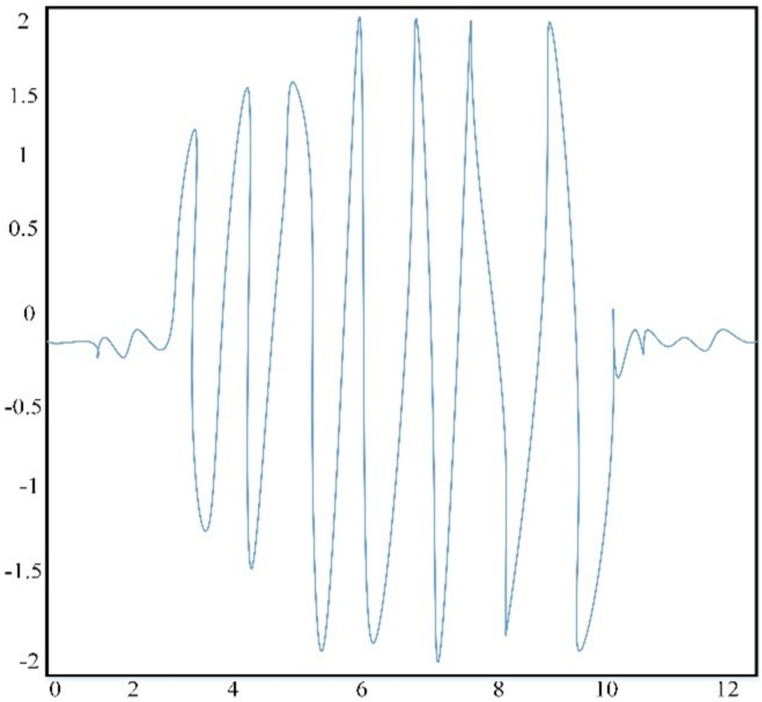

Fig. 8: Displacement over time

demonstrates a noteworthy improvement in the reduction of solution discrepancy. This suggests that the refinement or alteration introduced in Mesh 2 has a positive impact on aligning the numerical results more closely with the experimental values. However, the subsequent transitions from Mesh 2 to Mesh 5 (M2 to M5) exhibit a less substantial percentage reduction in the deviation. This implies that the further refinement or changes in mesh configurations beyond Mesh 2 do not significantly contribute to enhancing the agreement between numerical and experimental results. The differences observed in the results among Mesh 2, Mesh 3 (M3), Mesh 4 (M4), and Mesh 5 (M5) are deemed insignificant, indicating that these additional refinements in mesh configurations do not lead to a substantial improvement in solution accuracy.

This detailed analysis of the absolute deviation and the observed trends in solution discrepancies provides valuable insights into the effectiveness of different mesh schemes. It informs researchers about the critical points of improvement and guides the selection of an optimal mesh configuration for achieving better agreement with experimental data. The assessment of these results aids in refining the numerical model, ensuring that it accurately represents the physical behavior of the system under consideration.

7 Conclusion:

In conclusion, this study has presented a comprehensive investigation into the numerical simulation of fluid-structure interactions (FSI) on super-tall slender structures using advanced two-way coupling techniques. Through rigorous numerical simulations, research have gained valuable insights into the complex interplay between fluid flow dynamics and structural responses, which are crucial for the design and optimization of such structures in challenging wind environments. The results of research simulations have demonstrated the effectiveness of the proposed two-way coupling framework in capturing the intricate FSI phenomena. Research have observed detailed fluid flow patterns around the super-tall structures and analysed the corresponding structural deformations and dynamic responses. This comprehensive understanding of FSI behavior enhances research ability to predict and mitigate potential risks to structural integrity and occupant safety. However, it is essential to acknowledge certain limitations of research study. Firstly, while research numerical simulations provide valuable insights, they are inherently reliant on the accuracy of the mathematical models and numerical techniques employed. Further refinement and validation of these models against experimental data are necessary to improve confidence in the simulation results. Additionally, the computational resources required for conducting high-fidelity simulations of super-tall

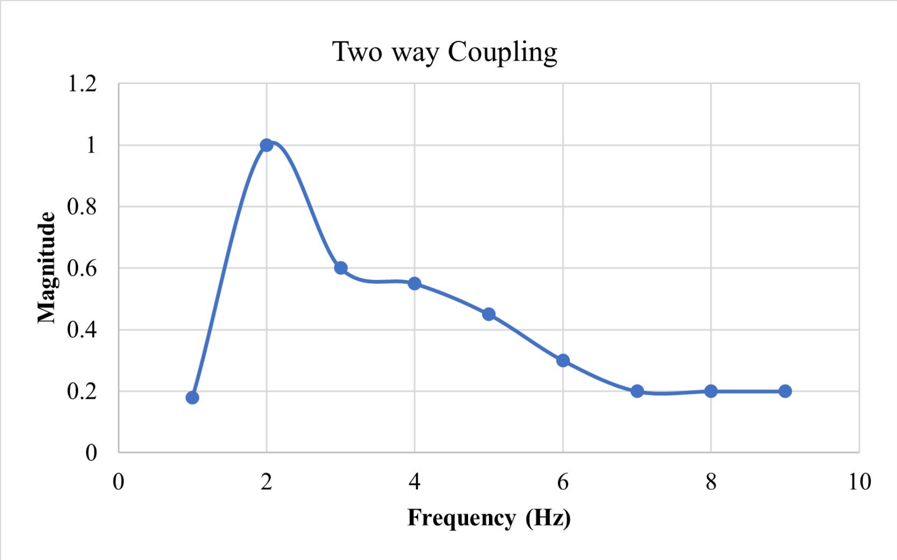

Fig. 9: Two Way Coupling

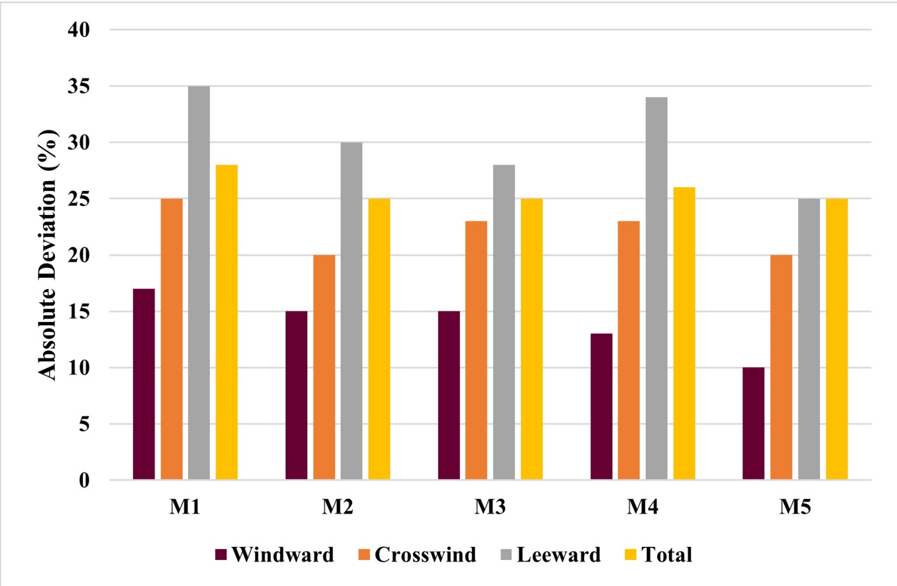

Fig. 10: Absolute Deviation

structures can be substantial, posing challenges in terms of computational cost and efficiency. To address these limitations and further advance the field, future research directions can focus on several aspects. Firstly, there is a need for continued development and validation of more sophisticated numerical models for capturing FSI effects with greater accuracy and efficiency. Additionally, experimental validation of the numerical results, perhaps through wind tunnel testing or full-scale monitoring of super-tall structures, can provide invaluable validation and calibration data. Furthermore, exploring advanced optimization techniques for structural design and control strategies to mitigate FSI-induced vibrations and dynamic instabilities represents an important avenue for future investigation. Overall, continued interdisciplinary research efforts combining computational modelling, experimental validation, and innovative design approaches are essential for advancing our understanding and management of FSI in super-tall slender structures.

Acknowledgments

The authors thank Prince Sattam bin Abdulaziz University for funding this research work through the project number (PSAU/2023/01/25218).

Conflicts of Interest

The author declares that he has no conflicts of interest to report regarding the present study.

References

[1] F. Ding and A. Kareem, Tall buildings with dynamic facade under winds, *Engineering* **6**(12) (2020) 1443–1453.

[2] M. L. Nehdi, Only tall things cast shadows: Opportunities, challenges and research needs of self-consolidating concrete in super-tall buildings, *Construction and Building Materials* **48** (2013) 80–90.

[3] J. Pitroda and J. Singh, Evolution of super tall and super slender skyscrapers structural systems in conjunction with architectural forms & aesthetics, in *Proceedings of International Conference on: Engineering: Issues, opportunities and Challenges for Development*, **9**2016, pp. 206–222.

[4] J. Szołomicki and H. Golasz-Szołomicka, The modern trend of super slender residential buildings, *Budownictwo i Architektura* **20**(1) (2021).

[5] F.-K. Benra, H. J. Dohmen, J. Pei, S. Schuster and B. Wan, A comparison of one-way and two-way coupling methods for numerical analysis of fluid-structure interactions, *Journal of applied mathematics* **2011**(1) (2011) p. 853560.

[6] G. Li, W. Li, F. Han, S. Lin and X. Zhou, Experimental investigation and one-way coupled fluid-structure interaction analysis of gas-liquid two-phase flow conveyed by a subsea rigid m-shaped jumper, *Ocean Engineering* **285** (2023) p. 115292.

[7] W. Liu, W. Luo, M. Yang, T. Xia, Y. Huang, S. Wang, J. Leng and Y. Li, Development of a fully coupled numerical hydroelasto-plastic approach for offshore structure, *Ocean Engineering* **258** (2022) p. 111713.

[8] P. Lakshmynarayanana and S. Hirdaris, Comparison of nonlinear one-and two-way ffsi methods for the prediction of the symmetric response of a containership in waves, *Ocean engineering* **203** (2020) p. 107179.

[9] E. B. Allen, T. M. Khoshgoftaar and Y. Chen, Measuring coupling and cohesion of software modules: an information-theory approach, in *Proceedings seventh international software metrics symposium*, IEEE2001, pp. 124–134.

[10] Y. Gao, L. He, Z. Jiang, J. Zhou, Y. Shi and W. Bai, Investigation of coupling effect between a unidirectional air waveguide and two cavities with one-way rotating state, *Optica Applicata* **50**(1) (2020) 49–59.

[11] P. Patel, H.-S. Im and G. Zha, Numerical investigation of non-synchronous vibration with fluid-structure interaction using delayed detached eddy simulation, in *AIAA Scitech 2020 Forum*, 2020, p. 0384.

[12] S. De, B. S. M. Ebna Hai, A. Doostan and M. Bause, Prediction of ultrasonic guided wave propagation in fluid–structure and their interface under uncertainty using machine learning, *Journal of Engineering Mechanics* **148**(3) (2022) p. 04021161.

[13] B. Yan, H. Ren, D. Li, Y. Yuan, K. Li, Q. Yang and X. Deng, Numerical simulation for vortex-induced vibration (viv) of a high-rise building based on two-way coupled fluid-structure interaction method, *International Journal of Structural Stability and Dynamics* **22**(03n04) (2022) p. 2240010.

[14] S. Huang, R. Li and Q. Li, Numerical simulation on fluid-structure interaction of wind around super-tall building at high reynolds number conditions, *Struct. Eng. Mech* **46**(2) (2013) 197–212.

[15] B. Yan, H. Ren, D. Li, Y. Yuan, K. Li, Q. Yang and X. Deng, Numerical simulation for vortex-induced vibration (viv) of a high-rise building based on two-way coupled fluid-structure interaction method, *International Journal of Structural Stability and Dynamics* **22**(03n04) (2022) p. 2240010.

[16] F.-K. Benra, H. J. Dohmen, J. Pei, S. Schuster and B. Wan, A comparison of one-way and two-way coupling methods for numerical analysis of fluid-structure interactions, *Journal of applied mathematics* **2011**(1) (2011) p. 853560.

[17] J.-M. Vassen, P. DeVincenzo, C. Hirsch and B. Leonard, Strong coupling algorithm to solve fluid-structure-interaction problems with a staggered approach, in *7th European symposium on aerothermodynamics*, **692**2011, p. 128.

[18] W. Cui and L. Caracoglia, A fully-coupled generalized model for multi-directional wind loads on tall buildings: A development of the quasi-steady theory, *Journal of Fluids and Structures* **78** (2018) 52–68.

[19] K. Wijesooriya, D. Mohotti, A. Amin and K. Chauhan, An uncoupled fluid structure interaction method in the assessment of structural responses of tall buildings, in *Structures*, **25**, Elsevier2020, pp. 448–462.

[20] A. Elshaer, *Aerodynamic optimization and wind load evaluation framework for tall buildings*, PhD thesis, The University of Western Ontario (Canada)2017.

[21] S. Huang, R. Li and Q. Li, Numerical simulation on fluid-structure interaction of wind around super-tall building at high reynolds number conditions, *Struct. Eng. Mech* **46**(2) (2013) 197–212.

[22] S. Leng, S.-W. Li, Z.-Z. Hu, H.-Y. Wu and B.-B. Li, Development of a micro-in-meso-scale framework for simulating pollutant dispersion and wind environment in building groups, *Journal of Cleaner Production* **364** (2022) p. 132661.