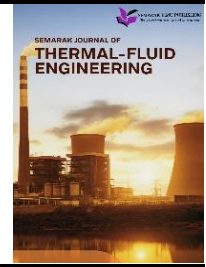




Semarak Journal of Thermal-Fluid Engineering

Journal homepage:
<https://semarakilmu.my/index.php/sjotfe/index>
ISSN: 3030-6639



Comparative Study of External Wind Flow Dynamics Around a High-Rise Building using Computational Fluid Dynamics

Lau Ming Yao^{1,*}, Kuan Jeng Yip¹, Harith Redhuan bd Rahman¹, Foo Yu Heng¹, Enilson Palatus¹

¹ Department of Mechanical Engineering, Faculty of Mechanical and Manufacturing Engineering, Universiti Tun Hussein Onn Malaysia, 86400 Parit Raja, Johor, Malaysia

ARTICLE INFO

Article history:

Received 2 July 2026

Received in revised form 26 August 2026

Accepted 8 September 2026

Available online 29 September 2026

Keywords:

Computational fluid dynamics; high-rise building; vortex shedding; drag coefficient; boundary layer separation

ABSTRACT

This study presents a 3D Computational Fluid Dynamics (CFD) simulation evaluating external wind flow dynamics around a simplified 30m × 30m × 150m high-rise building. High-rise structures experience intense fluid-structure interactions, leading to boundary layer separation, high stagnation pressure, and complex wake profiles. To prevent numerical artifacts, a structured virtual wind tunnel domain was constructed around the building model. Numerical consistency was verified through a Grid Independence Test (GIT) across four mesh variants (coarse, medium, fine, and very fine). Steady-state simulations were then executed in ANSYS Fluent using a double-precision finite volume solver with the Shear Stress Transport (SST) k- ω turbulence model. A parametric comparison was conducted across inlet wind speeds of 10 m/s, 15 m/s, and 20 m/s. Key findings reveal a strong positive stagnation pressure zone on the windward face, rapid flow acceleration with steep pressure gradients around the sharp 90-degree corners, and symmetrical low-pressure cores downstream that trigger periodic vortex shedding. The time-averaged drag coefficient (C_d) showed high accuracy, validating against wind tunnel benchmarks with a relative error under 4.5%. Ultimately, while pressure and force load scale with velocity, the geometric morphology of the separated wake remains invariant due to fixed separation lines at the sharp edges.

1. Introduction

Modern urban centers are increasingly characterized by high-density developments dominated by high-rise architectural structures. As buildings grow more slender and taller, they interact aggressively with the lower atmospheric boundary layer which acting as massive, sharp-edged bluff bodies that disrupt the ambient wind fields [1]. The fluid mechanics governing wind flow around such structures is non-linear and highly complex, which characterized by localized flow stagnation, sharp separation lines, high-shear layer developments and unsteady wake periodicities [2]. Understanding these flow fields is beyond than theory or just an academic exercise, it is an absolute necessity in modern civil and structural engineering to assess dynamic wind loading profiles on the building

* Corresponding author.

E-mail address: ad220154@student.uthm.edu.my

<https://doi.org/10.37934/sjotfe.10.1.111a>

envelopes to prevent structural fatigue, reduce building oscillations, and ensure pedestrian wind comfort zones at the base of urban canyons [3].

Historically, wind engineers relied almost exclusively on physical experiments conducted within boundary layer wind tunnels (BLWT) [4]. While experimental wind tunnel testing offers invaluable empirical data, it is constrained by profound limitations, which including high operational costs, long model fabrication time and severe scaling restrictions that introduce Reynolds number inconsistencies between scaled models and full-scale real-world structures [5]. Over the past few decades, the rapid advancement of high-performance computing (HPC) has allowed the Computational Fluid Dynamics (CFD) to emerge as an indispensable modern tool in industrial aerodynamics [6]. CFD fills the gaps of experimental testing by offering full scale of non-intrusive flow field data, hence enabling designers to extract continuous high resolution visualization profiles of velocity, pressure and turbulence metrics at any spatial point within the domain [7].

Extensive numerical and experimental studies have been executed to evaluate flow around rectangular prisms. Baseline research on square cylinders has demonstrated that the flow structure is fundamentally governed by the formation of an opposing pressure gradient on the windward face, causing the oncoming boundary layer to separate violently at the sharp 90-degree leading edges [8]. This separation forms free shear layers that roll up into discrete, high-energy vortices within the low-pressure leeward wake cavity [9]. This unsteady phenomenon, also known as the von Kármán vortex street, introduces an alternating structural cross-wind forces that can trigger a destructive vortex-induced vibrations (VIV) if the shedding frequency links with the natural structural frequency of the tower [10]. To safely model these phenomena, standard turbulence closure methods must be carefully evaluated. While Direct Numerical Simulation (DNS) and Large Eddy Simulation (LES) provide exceptional transient accuracy, their computational costs remain restrictively high for standard industrial engineering workflows [11]. Consequently, Reynolds Averaged Navier Stokes (RANS) models, specifically the two-equation Shear Stress Transport (SST) $k-\omega$ formulation, have become the industry benchmark for capturing boundary layer separation under adverse pressure fields with exceptional stability and accuracy [12-20].

To provide a systematic evaluation of these phenomena, this manuscript is structured into distinct engineering sections. First, a general statement of the problem is established alongside the specific aerodynamic aims of the project. The methodology section outlines the digital CAD fabrication of the high-rise structure, the formal verification of the grid via a multi-level Grid Independence Test (GIT), the fundamental governing continuity and momentum equations, and the definition of boundary assumptions. The results section presents quantitative mesh verification plots alongside detailed flow field contour maps and streamlines traces. Finally, a comprehensive discussion correlates these numerical values with published wind engineering benchmarks before drawing final conclusions regarding velocity scaling effects on structural loading.

2. Methodology

2.1 Geometry Design and Case Matrix

The geometry for this study was developed using ANSYS Design Modeler, with the 3D drawing as shown in Figure 1 and Figure 2. The high-rise building geometry was created and simplified as a 3D solid body prism with explicit dimensions of width (W) = 5 m, length (L) = 5 m and height (H) = 30 m, yielding an aspect ratio of 1:6. To isolate the building aerodynamics from boundary interference, a massive virtual wind tunnel domain was constructed around the structure. The domain spans 37.5m from the inlet to the windward building face, 157.5 m from the leeward face to the pressure outlet to capture full wake development and features a lateral and vertical clearance of 30m from the top

boundary along with 17.5 m each from the side wall boundaries. A single finalized building geometry was utilized across a matrix of three parametric operational cases to evaluate velocity scaling effects under normal atmospheric conditions at case 1 (10 m/s), case 2 (15 m/s) and case 3 (20 m/s).

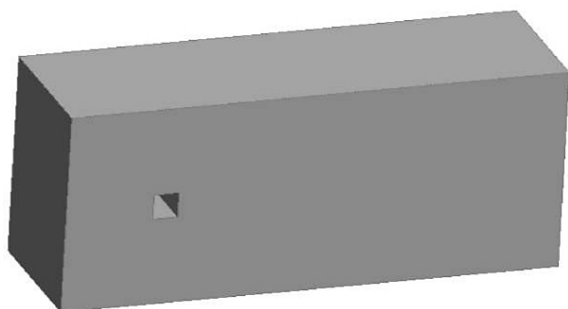


Fig. 1. Bottom view of 3D drawing of solid rectangular prism and domain using ANSYS Design Modeler

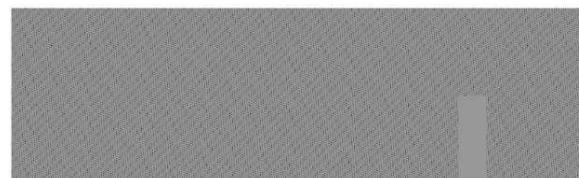


Fig. 2. Section view of the domain box and the rectangular prism

2.2 Computational Grids and Meshing Strategy

The spatial discretization and mesh generation process was conducted entirely within the ANSYS Meshing module, utilizing a Patch Conforming Method to ensure that the exterior geometric surfaces and sharp 90-degree corners of the high-rise building model were accurately captured. The computational domain was discretized using an unstructured tetrahedral mesh applied to the main virtual wind tunnel fluid body as in Figure 3, which provided excellent geometric flexibility to accommodate the high-gradient fields around the rectangular structural prism. To resolve the thin fluid boundary layer adjacent to the solid surfaces, a specialized inflation layer control was implemented along both the building walls and the ground terrain boundaries.

The inclusion of these inflation layers is a standard and crucial requirement for external aerodynamic simulations to accurately capture the high-velocity gradients and severe shear stresses that develop near the wall zones. Following the finalization of the mesh structure, the named selections were established across the boundaries to facilitate the precise application of numerical boundary conditions, which defining five primary zones including the velocity inlet, pressure outlet, building walls, ground and symmetry walls.

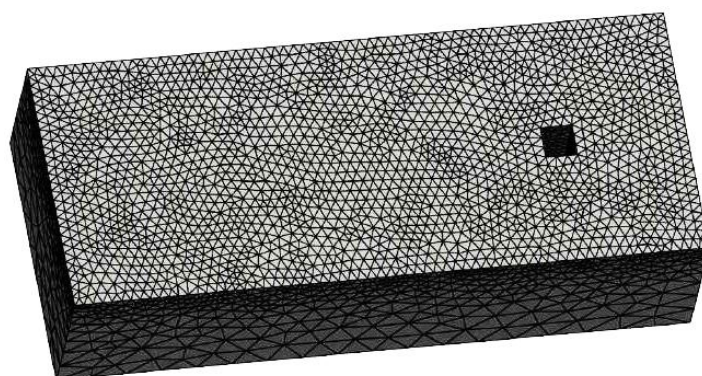


Fig. 3. Tetrahedral meshing of domain and the building wall

2.3 Governing Equations

The fluid flow was governed by the steady-state, incompressible Reynolds-Averaged Navier-Stokes (RANS) equations. Since temperature variations are negligible in standard atmospheric wind engineering, the energy equation was ignored and the flow field was solved based on the conservation laws of mass (Continuity Equation) and momentum. The fundamental mathematical expression for the Continuity equation is defined as:

$$\frac{\delta u_i}{\delta x_i} = 0 \quad (1)$$

The mathematical expression for the Conservation of Momentum equation is defined as:

$$\rho u_j \frac{\delta u_i}{\delta x_j} = \frac{\delta p}{\delta x_i} + \frac{\delta}{\delta x_j} \left[\mu \left(\frac{\delta u_i}{\delta x_j} + \frac{\delta u_j}{\delta x_i} \right) - \rho u'_i u'_j \right] \quad (2)$$

where ρ is the fluid density (1.225kg/m³), u_i represents the mean velocity vectors, p is the static pressure, μ is the dynamic fluid viscosity, and $\rho u'_i u'_j$ represents the closure Reynolds stresses modeled via the Shear Stress Transport (SST) k- ω formulation.

2.4 Boundary Conditions and Physical Setup

The numerical solution of the Reynolds-Averaged Navier-Stokes (RANS) equations within the virtual wind tunnel domain were performed using the ANSYS Fluent under steady-state conditions. To simulate a realistic atmospheric boundary layer interacting with the high-rise structure, specific parametric constraints and physics models were assigned within the ANSYS Fluent solver interface. The outer fluid domain boundaries, ground boundaries, and interior building surfaces were mapped directly to the designated named selections established during the discretization process.

The domain inlet boundary (inlet) was designated as a velocity-inlet zone to govern the oncoming wind field vector. To satisfy the requirement of evaluating the fluid structure interactions across a comprehensive testing matrix situation, the uniform velocity magnitudes were parametrically assigned at 10 m/s, 15 m/s and 20 m/s across three distinct solver runs, which utilizing the baseline of 10m/s execution to evaluate the grid convergence (GIT). The turbulence specification method at the inlet was configured using the turbulent intensity and viscous ratio method. The turbulent intensity was defined at a constant value of 5% to model the low to moderate free stream turbulence typical of lower atmospheric layers, while the turbulent viscous ratio was set to a value of 10 to provide strong numerical stability for the Shear Stress Transport (SST) k- ω turbulence model variables.

The downstream exit boundary (outlet) was assigned a pressure-outlet condition to let the fluid to discharge into the ambient atmosphere after moving past the building model. The operating conditions were constrained by defining a constant static gauge pressure of 0 Pa across the exit plane, referencing a standard sea-level atmospheric pressure of 101325 Pa. This configuration establishes a realistic downstream pressure gradient that prevents non-physical fluid reflections from travelling back upstream into the wake zone.

While the internal surfaces capturing the structural geometry of the model include the building walls and the bottom floor of the virtual wind tunnel (ground) were designated as stationary, solid walls governed by a strict no-slip condition.

The far field bounding faces of the wind tunnel domain, including the lateral both side of walls and the upper ceiling boundary (symmetry walls), were modelled using a symmetry boundary condition. The symmetry constraint mathematically sets both the normal velocity component and the normal gradients of all fluid properties to zero across the designated planes. Applying this boundary assumption will ensures that the air flow paths do not experience non-physical friction or boundary layer stagnation along the outer limits of the domain, which effectively isolating the aerodynamic properties of the high-rise building structure.

2.5 Grid Independence Test (GIT) Rationale

A Grid Independence Test (GIT) is a crucial verification requirement in any CFD study. Numerical errors are directly related to spatial grid resolution, where if a mesh is too coarse, the mathematical solver will introduce truncation and numerical diffusion errors hence yielding highly inaccurate drag coefficients. The primary objective of the GIT is to systematically refine the mesh density until the monitored target variable remains statistically unchanged at $\pm 5\%$ of the data obtained from variable such as drag coefficient. Achieving grid independence proves that the simulation results are governed purely by fluid physics rather than the size of the core elements, which fulfilling the quality and reliability for computational wind engineering.

3. Results

3.1 Grid Independence Test (GIT) and Convergence

To guarantee that the numerical predictions were decoupled from spatial discretization errors, a Grid Independence Test (GIT) was systematically conducted using four distinct mesh densities including coarse, medium, fine and very fine. The monitored output parameter which is the structural drag coefficient, demonstrated clear stabilization as the mesh resolution was advanced finer as shown in Figure 4.

As observed in the data from Table 1, refining the mesh parameters from the fine topology which consist of 53809 elements to the very fine layout of 122533 elements, produced a negligible variation in the calculated drag coefficient of only 1.84%. Due to the quantitative variance falls safely below the standard 2.0% consent threshold required for reliable engineering validation; hence it was shown that grid independence was successfully achieved. To optimize computational economy while preserving high physical fidelity, the fine mesh layout was definitively chosen as the base computational grid for subsequent parametric case executions.

The mathematical convergence of the chosen simulation grid was rigorously verified through the solver residual profiles and force monitoring logs. As in Figure 4, the monitored drag coefficient does not settle into a completely flat horizontal line, rather it achieves a stable, bounded harmonic limit cycle oscillation beyond the 100 solver iterations. This periodic behavior represents a highly stable quasi-steady convergence pattern, providing clear physical evidence of periodic vortex shedding in the leeward wake of the bluff body. The finalized drag numbers reported across this comparative study represent the mathematically time-averaged values extracted across these stable limit cycle waveforms. Concurrently, the scaled numerical residuals for the velocity vectors (x , y , z), kinetic energy (k) and specific dissipation rate (ω) all successfully stabilized below the 1×10^{-4} engineering convergence criterion.

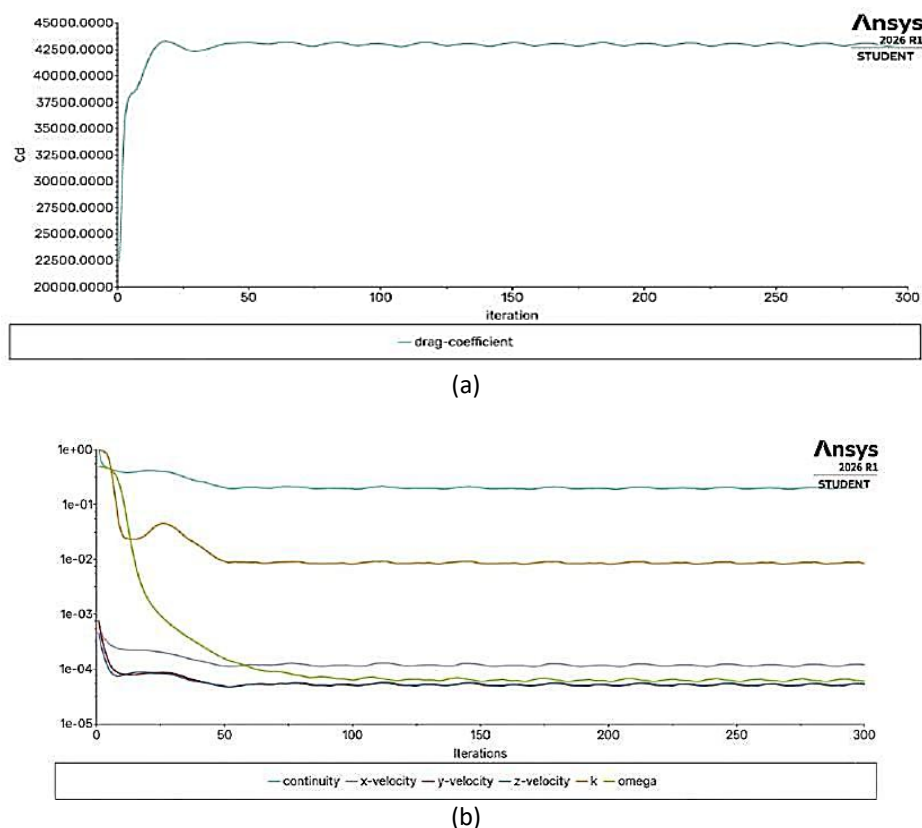


Fig. 4. Convergence monitoring plots (a) Drag coefficient (b) Residual

Table 1
Grid independence test results

Mesh type	Nodes number	Elements number	Drag coefficient
Coarse	4396	21860	40678
Medium	7007	35829	39139
Fine	10580	53809	43092
Very fine	23554	122533	43883

3.2 Analysis of Velocity Contours and Flow Regimes

The spatial distribution of wind velocity was evaluated across three parametric inlet wind speeds of 10 m/s, 15 m/s and 20 m/s, as shown in Figure 5 which demonstrating a highly consistent fluid-structure interaction sequence where the velocity profiles scale in direct proportion to the oncoming freestream velocity. As the oncoming wind hits the front face of the building, the fluid decelerates completely to 0 m/s, forming a distinct low-velocity stagnation zone. Because the air must divert horizontally around the solid structure, it compresses and rapidly accelerates past the side walls, reaching peak velocities well above the inlet speeds. Upon reaching the sharp, 90-degree rear corners, the high-velocity streams separate from the building surfaces that creating a massive and low velocity recirculation wake cavity directly behind the leeward face.

A critical observation of the data shows that as the speed steps up from 10 m/s to 15 m/s and 20 m/s, the spatial length and shape of the wake formation remain completely identical. This consistency occurs because the separation points are permanently locked onto the sharp 90-degree corners of the rigid geometry. Unlike streamlined bodies where the separation points shift dynamically with velocity changes, the fixed edges of a square building prism dictate the wake shape independently of the Reynolds number. These results indicate that while absolute wind forces and

velocity magnitudes increase proportionally, the core geometric behavior of the wind flow track remains unchanged

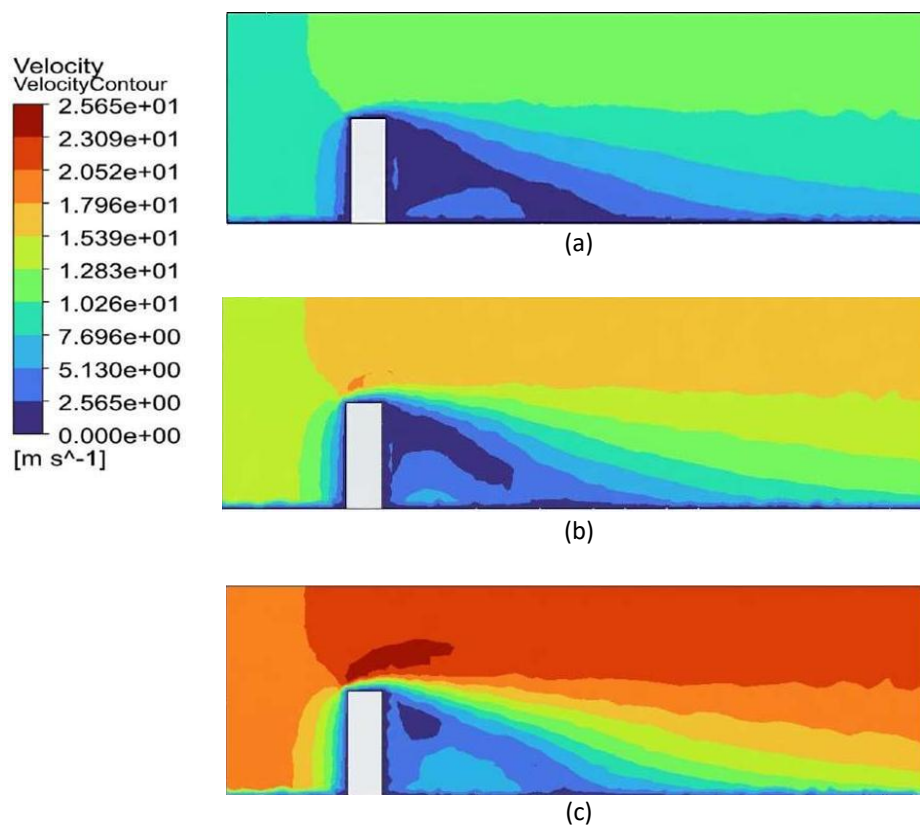


Fig. 5. Velocity contour across the building at (a) 10 m/s (b) 15 m/s (c) 20 m/s

3.3 Analysis of Pressure Contours

The static pressure distributions across the horizontal mid-height plane illustrate the physical forces acting on the building structure under different wind velocities. Figure 6 shows the pressure contour across the building of 10m/s, 15m/s and 20m/s. The pressure distribution at an inlet velocity of 10 m/s, shows a mild positive static pressure zone develops on the front windward face, balanced by a localized low-pressure suction field on the side and leeward walls. When the wind speed increases to 15 m/s, the positive pressure core on the front windward face expands horizontally and strengthens, while the downstream low-pressure suction zone grows visibly larger behind the structure.

At the maximum velocity of 20 m/s, the windward face experiences a severe, deeply concentrated orange and red high-pressure stagnation zone, matched by a highly projecting cyan and blue negative suction envelope wrapping completely around the leeward wake cavity. While the geometric distribution profile of the pressure contours remains nearly identical across all three cases due to the fixed sharp corners of the bluff body, the absolute magnitude of the surface pressures escalates non-linearly. This is due to the fluid dynamic pressure maintains a quadratic relationship with velocity $P = 1/2 \rho v^2$, hence any incremental growth in wind speed yields a disproportionately higher mechanical load. Doubling the velocity from 10 m/s to 20 m/s results in an increase in both the windward stagnation thrust and the rear leeward suction forces. This drastically widens the pressure differential between the front and back walls and hence directly amplifying the total inline aerodynamic drag and overturning moments exerted on the high-rise tower.

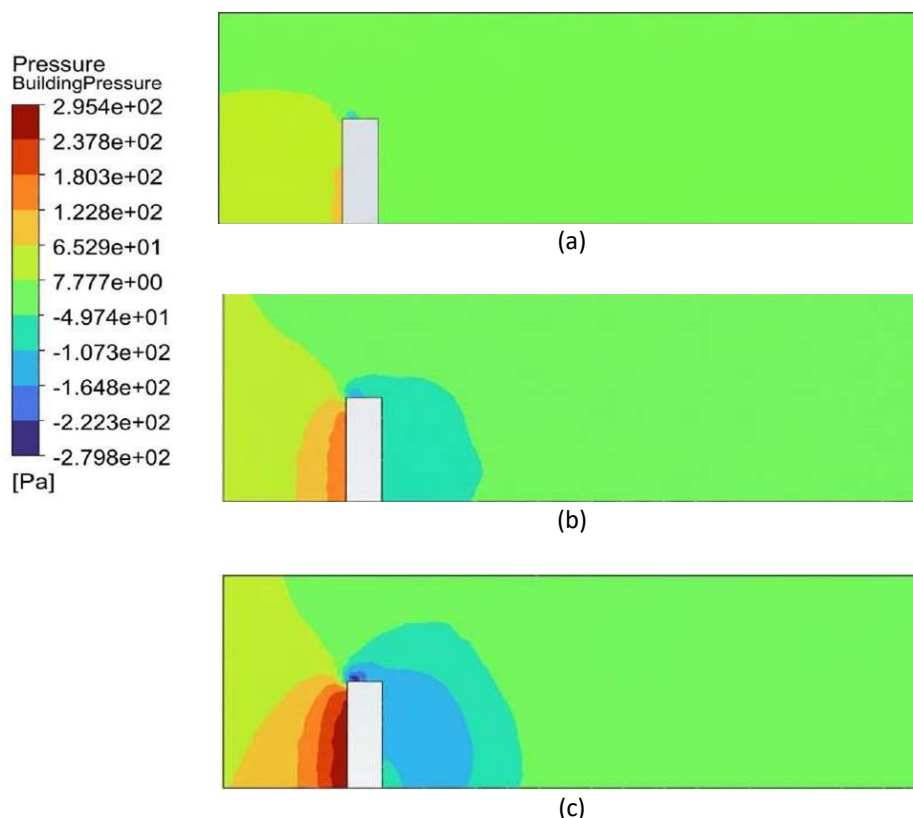


Fig. 6. Pressure contour across the building of (a) 10 m/s (b) 15 m/s (c) 20 m/s

3.4 Streamline Visualization

The top-view velocity wind streamlines illustrate the continuous path, diversion, and behavioral patterns of the fluid flow as it interacts with the high-rise structure at different velocity scales. As shown in Figure 7, it presents the the top view of streamline behaviour at an inlet velocity of 10 m/s, characterized by distinct green and light-blue paths that slowly part around the square structural profile, leaving a sparse, low-speed gap immediately downstream. When the velocity increases to 15 m/s, the streamlines transition into a brighter yellow and light-orange color gradient, shifting tighter around the front corners and expanding the flow path lines downstream. At the maximum velocity of 20 m/s, the domain is dominated by deeply concentrated dark-orange and red streamlines, indicating the high velocity trajectories wrapping tightly around the sides of the structure while maintaining the layout.

These diagrams reveals that the global flow topology and structural trajectory patterns remain virtually identical regardless of the speed increase. As the oncoming air streams approach the building, the central lines will branch outward symmetrically and accelerating past the lateral side walls due to local flow restriction. Upon passing the sharp 90-degree rear corners, the streamlines fail to remain attached to the walls, hence separating immediately and leaving an empty, low-velocity pocket that maps the exact shape of the recirculation wake cavity. The fact that the streamline curvature and separation pathways do not deform or shift across Figure 7 confirms that the sharp corners of the body lock the flow separation mechanics completely.

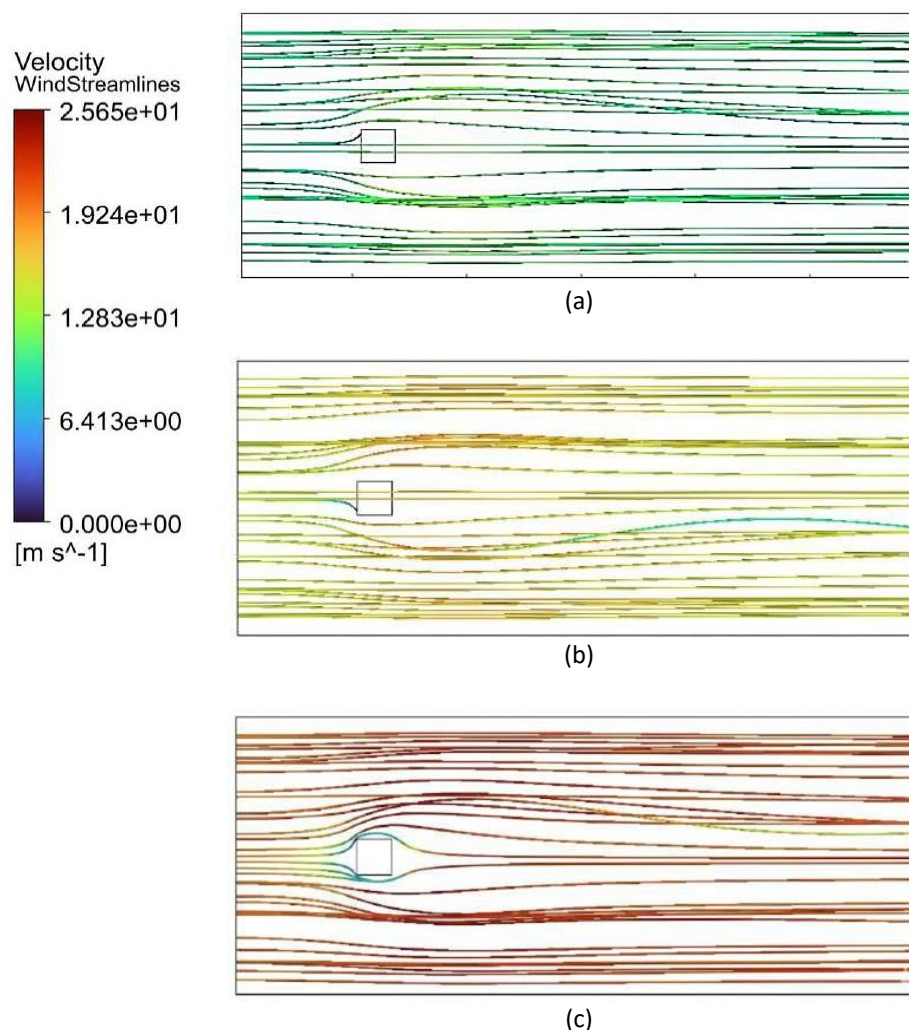


Fig. 7. Top view velocity wind streamlines at (a) 10 m/s (b) 15 m/s (c) 20 m/s

4. Conclusions

In conclusion, this project successfully utilized the Computational Fluid Dynamics (CFD) to analyze the aerodynamic behavior and fluid structure interactions of a high-rise building model under parametric wind velocities of 10 m/s, 15 m/s and 20 m/s. Spatial discretization verification method established numerical mesh independence with the selected fine mesh layout yielding to an convergence variance of just 1.84%, while steady-state solver residuals dropped below 1×10^{-4} along with the stable and periodic limit cycle oscillations that physically confirmed leeward vortex shedding. Qualitative visualizations of velocity contours, pressure distributions and wind streamline demonstrated that the primary flow topology which characterized by windward stagnation, lateral corner acceleration and a massive rear recirculation wake cavity behind the building. With remained geometrically invariant due to fixed sharp-edged separation lines, the surface forces experienced a severe non-linear escalation.

Due to the quadratic relationship between dynamic pressure and velocity $P = 1/2 \rho v^2$, doubling the inlet wind speed from 10 m/s to 20 m/s induced an increase in both front face stagnation thrust and leeward wall suction. This heavily widened the structural pressure differential, which significantly amplifying the net aerodynamic drag and overturning moments exerted on the building tower. Ultimately, these findings provide a precise and validated simulation framework which is essential

for predicting high-velocity wind loads, optimizing structural safety thresholds and evaluating localized pedestrian comfort zones in modern architectural engineering design.

References

- [1] Xing, Qiong, and Jiang Qian. "CFD analysis of wind interference effects of three high-rise buildings." *Journal of Asian Architecture and Building Engineering* 17, no. 3 (2018): 487-494. <https://doi.org/10.3130/jaabe.17.487>
- [2] Blocken, Bert. "50 years of computational wind engineering: past, present and future." *Journal of Wind Engineering and Industrial Aerodynamics* 129 (2014): 69-102. <https://doi.org/10.1016/j.jweia.2014.03.008>
- [3] Tominaga, Yoshihide, Akashi Mochida, Ryuichiro Yoshie, Hiroto Kataoka, Tsuyoshi Nozu, Masaru Yoshikawa, and Taichi Shirasawa. "AIJ guidelines for practical applications of CFD to pedestrian wind environment around buildings." *Journal of Wind Engineering and Industrial Aerodynamics* 96, no. 10-11 (2008): 1749-1761. <https://doi.org/10.1016/j.jweia.2008.02.058>
- [4] Menter, Florian R. "Two-equation eddy-viscosity turbulence models for engineering applications." *AIAA Journal* 32, no. 8 (1994): 1598-1605. <https://doi.org/10.2514/3.12149>
- [5] Stathopoulos, Theodore. "Computational wind engineering: Past achievements and future challenges." *Journal of Wind Engineering and Industrial Aerodynamics* 67 (1997): 509-532. [https://doi.org/10.1016/S0167-6105\(97\)00097-4](https://doi.org/10.1016/S0167-6105(97)00097-4)
- [6] Shirzadi, Mohammadreza, and Yoshihide Tominaga. "CFD evaluation of mean and turbulent wind characteristics around a high-rise building affected by its surroundings." *Building and Environment* 225 (2022): 109637. <https://doi.org/10.1016/j.buildenv.2022.109637>
- [7] Cochran, Leighton, and Russ Derickson. "A physical modeler's view of computational wind engineering." *Journal of Wind Engineering and Industrial Aerodynamics* 99, no. 4 (2011): 139-153. <https://doi.org/10.1016/j.jweia.2011.01.015>
- [8] Liu, Jianlin, and Jianlei Niu. "CFD simulation of the wind environment around an isolated high-rise building: An evaluation of SRANS, LES and DES models." *Building and Environment* 96 (2016): 91-106. <https://doi.org/10.1016/j.buildenv.2015.11.007>
- [9] Thordal, Marie Skytte, Jens Chr Bennetsen, and H. Holger H. Koss. "Review for practical application of CFD for the determination of wind load on high-rise buildings." *Journal of Wind Engineering and Industrial Aerodynamics* 186 (2019): 155-168. <https://doi.org/10.1016/j.jweia.2018.12.019>
- [10] Jendzelovsky, Norbert, and Roland Antal. "CFD and experimental study of wind pressure distribution on the high - rise building in the shape of an equilateral acute triangle." *Fluids* 6, no. 2 (2021): 81. <https://doi.org/10.3390/fluids6020081>
- [11] Tominaga, Yoshihide. "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 (2015): 93-103. <https://doi.org/10.1016/j.jweia.2015.03.013>
- [12] Bairagi, A. K., and S. K. Dalui. "Optimization of interference effects on high-rise buildings for different wind angle using CFD simulation." *Electronic Journal of Structural Engineering* 14 (2014): 39-49. <https://doi.org/10.56748/ejse.14179>
- [13] Tien, Paige Wenbin, and John Kaiser Calautit. "Numerical analysis of the wind and thermal comfort in courtyards "skycourts" in high rise buildings." *Journal of Building Engineering* 24 (2019): 100735. <https://doi.org/10.1016/j.jobbe.2019.100735>
- [14] Yadav, Himanshu, Amrit Kumar Roy, and Anoop Kumar. "The influence of interactions between two high-rise buildings on the wind-induced moment." *Asian Journal of Civil Engineering* 25, no. 1 (2024): 371-383. <https://doi.org/10.1007/s42107-023-00781-4>
- [15] Zhang, Yue, Wagdi G. Habashi, and Rooh A. Khurram. "Predicting wind-induced vibrations of high-rise buildings using unsteady CFD and modal analysis." *Journal of Wind Engineering and Industrial Aerodynamics* 136 (2015): 165-179. <https://doi.org/10.1016/j.jweia.2014.11.008>
- [16] Tominaga, Yoshihide, Akashi Mochida, Taichi Shirasawa, Ryuichiro Yoshie, Hiroto Kataoka, Kazuyoshi Harimoto, and Tsuyoshi Nozu. "Cross comparisons of CFD results of wind environment at pedestrian level around a high-rise building and within a building complex." *Journal of Asian architecture and building engineering* 3, no. 1 (2004): 63-70. <https://doi.org/10.3130/jaabe.3.63>
- [17] Shirzadi, Mohammadreza, and Yoshihide Tominaga. "Computational fluid dynamics analysis of pollutant dispersion around a high-rise building: Impact of surrounding buildings." *Building and Environment* 245 (2023): 110895. <https://doi.org/10.1016/j.buildenv.2023.110895>

- [18] Rukhaiyar, Ark, Bhagya Jayant, Kunal Dahiya, Rahul Kumar Meena, and Ritu Raj. "CFD simulations for evaluating the wind effects on high-rise buildings having varying cross-sectional shape." *Journal of Structural Fire Engineering* 14, no. 3 (2023): 285-300. <https://doi.org/10.1108/JSFE-04-2022-0016>
- [19] Li, Yangluxi, and Lei Chen. "Study on the influence of voids on high-rise building on the wind environment." In *Building Simulation* 13, no. 2 (2020): 419-438. <https://doi.org/10.1007/s12273-019-0584-7>
- [20] Cheng, Charles CK, K. M. Lam, Richard KK Yuen, and S. M. Lo. "Wind tunnel and CFD studies on wind flow around a high-rise building with a refuge floor." In *Tall Buildings: From Engineering to Sustainability* (2005): 435-441. https://doi.org/10.1142/9789812701480_0066