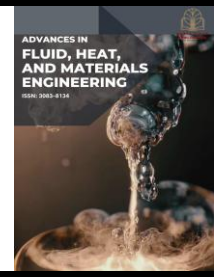




# Advances in Fluid, Heat and Materials Engineering

Journal homepage:  
<https://karyailham.com.my/index.php/afhme/index>  
ISSN: 3083-8134



## CFD Study of Wind Flow Interaction between Single and Double Building Blocks

Muhammad Hamizan Azli<sup>1,\*</sup>, Muhammad Iqbal Zahin Arsam<sup>1</sup>, Muhammad Iqram Hafiz Rusnizam<sup>1</sup>, Muhammad Izzul Afif Mohd Nazaruddin<sup>1</sup>, Nur Izzat Naquiuddin Nur Hisham<sup>1</sup>

<sup>1</sup> Department of Mechanical and Manufacturing Engineering, Universiti Tun Hussein Onn Malaysia, 86400 Parit Raja, Johor, Malaysia

### ARTICLE INFO

#### Article history:

Received 12 January 2026  
Received in revised form 18 February 2026  
Accepted 9 March 2026  
Available online 31 March 2026

#### Keywords:

Computational fluid dynamics; wind flow around; building interaction; single and double building blocks; urban wind environment; flow pattern; velocity field

### ABSTRACT

Wind flow around buildings is a critical factor in urban planning and wind engineering, as it directly affects pedestrian comfort and structural safety. When wind interacts with structures, it creates complex phenomena such as flow separation, recirculation zones, and wake formation. This study investigates the external flow behaviour around single and double building blocks using Computational Fluid Dynamics (CFD). Three-dimensional cubic models with dimensions of 20 mm × 20 mm × 20 mm were created and simulated using ANSYS Fluent. Air was selected as the working fluid, and three inlet velocities of 2 m/s, 5 m/s, and 8 m/s were applied to represent typical Malaysian wind conditions. Turbulent flow conditions were modelled using the standard k-epsilon turbulence model. A grid independence test was conducted to ensure that the results were independent of mesh resolution, leading to the selection of an 80,000 nodes mesh. Velocity and pressure contours were analysed to compare flow distribution and wake development between the two configurations. The results show that building arrangement significantly influences flow behaviour. The single-block case produced a predictable wake region, while the double-block configuration exhibited a distinct "channelling effect," where the wind accelerated significantly in the gap between the buildings. Higher inlet velocities intensified these patterns, leading to larger recirculation zones and merged wakes. In conclusion, the presence of neighbouring buildings and higher wind speeds creates more complex aerodynamic interactions that must be considered in urban design. These findings can help improve building layouts for better wind environmental quality in urban areas.

## 1. Introduction

Wind flow around buildings is an important concern in wind engineering and urban studies, especially in Malaysia where wind conditions affect comfort and building performance. As wind passes a building, it separates and forms recirculation and wake regions that influence wind loads, pedestrian comfort, and natural ventilation [1]. Understanding this behaviour is therefore essential for effective building design and urban planning. Computational Fluid Dynamics is widely used to

\* Corresponding author.

E-mail address: [dd220028@student.uthm.edu.my](mailto:dd220028@student.uthm.edu.my)

<https://doi.org/10.37934/afhme.8.1.3136a>

study wind building interaction because it provides detailed flow information at lower cost than wind tunnel testing [2].

Previous studies have mainly focused on wind flow around a single isolated building to understand basic aerodynamic behaviour [3]. However, real urban environments consist of multiple buildings, where interaction between structures significantly alters flow patterns, wake formation, and wind speed. Studies involving two or more buildings show that analysing a single building alone may not adequately represent actual urban wind conditions [4].

Wind velocity is another key factor affecting airflow around buildings. In Malaysia, wind speeds are generally moderate and vary due to monsoon seasons, terrain, and urban development. Wind speeds of 2 m/s, 5 m/s, and 8 m/s are commonly used in local CFD studies to represent typical urban wind conditions [5]. Despite this, many studies apply only one inlet wind speed, limiting their ability to represent realistic wind variations.

The main limitation in existing research is the lack of systematic comparison of wind flow behaviour when both building configuration and wind speed are varied together [6]. This reduces the applicability of CFD results for urban wind studies and preliminary building design in the Malaysian context [7]. Therefore, this study aims to investigate wind flow interaction around single and double building blocks using Computational Fluid Dynamics. Flow patterns and velocity fields are analysed under wind speeds of 2 m/s, 5 m/s, and 8 m/s to represent common Malaysian conditions, allowing comparison of airflow behaviour between single and double building configurations [8].

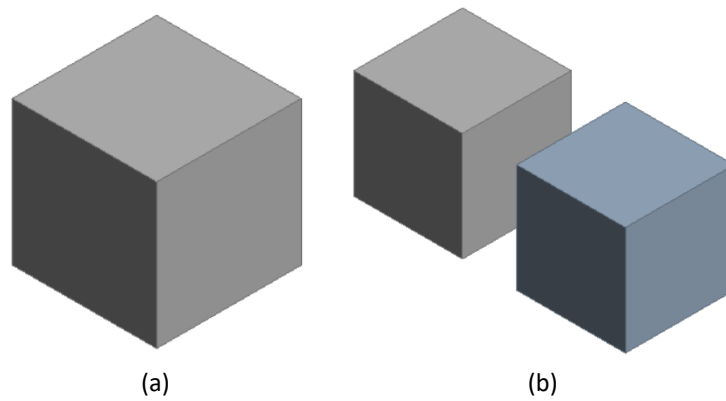
## **2. Methodology**

### **2.1 Design of Building Block**

In this study, a simplified building block geometry with dimensions of 20 mm × 20 mm × 20 mm is used to represent a typical mid-rise building in an idealised form. A cubic geometry is selected to reduce geometric complexity and allow clearer observation of wind flow behaviour around the structure [9]. This type of simplified building block is commonly adopted in CFD wind studies to focus on flow patterns and velocity distribution rather than architectural details. The building height is taken as the reference length scale for the analysis [10].

For the double building configuration, the spacing between the two building blocks is set to 0.5H, where H represents the building height of 20 m. This spacing corresponds to a centre-to-centre separation that allows noticeable aerodynamic interference between the buildings while avoiding complete blockage of the flow [11]. A spacing ratio of 0.5H is widely used in previous CFD and wind engineering studies to investigate wake interaction and flow acceleration effects between adjacent buildings [12]. This spacing is suitable for examining how the presence of a neighbouring building modifies the wind flow pattern and velocity field compared to a single building case.

Figure 1(a) show the design for single building block while the Figure 1(b) shows the double building block. The distance between the two building blocks in the double building arrangement is set to 0.5H, where H is the building height of 20 m, giving a spacing of 10 m between the blocks. This distance is chosen so that the two buildings are close enough to influence each other, but not so close that the flow is completely blocked. At this spacing, the wake formed behind the first building can interact directly with the second building, causing noticeable changes in the airflow between them. The wind tends to accelerate through the gap, while recirculation regions can form near the building surfaces. This makes the selected spacing suitable for observing how the presence of a nearby building affects the flow pattern and velocity field compared to a single building case.



**Fig. 1.** Design (a) Single building block (b) Double building block

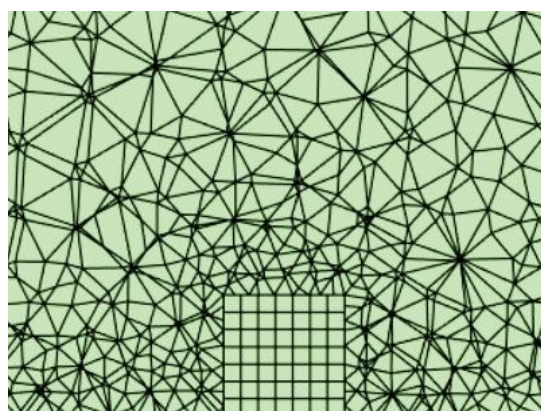
## 2.2 Discretization

The computational domain is divided into small control volumes so that the flow equations can be solved numerically. A finer mesh is applied near the building surfaces and wake regions to capture changes in velocity and flow separation more accurately. Coarser elements are used farther from the buildings where the flow is more uniform. This mesh arrangement helps achieve reliable results while keeping the computational cost reasonable [13].

### 2.2.1 Generate mesh

The mesh generation was performed using the tetrahedron method, which is well-suited for capturing the curved surfaces of the pipe and the geometry of the T-junction. Body sizing techniques were applied to control the element size throughout the computational domain.

Figure 2 shows the mesh of fluid domain. The computational domain is meshed using the tetrahedral method, which allows flexible discretization of complex flow regions around the building geometry. Tetrahedral elements are well suited for capturing irregular flow features such as separation and recirculation near sharp edges and corners. A finer tetrahedral mesh is applied close to the building surfaces to improve accuracy in predicting velocity gradients and wake formation. Coarser elements are used farther away from the building to reduce computational cost while maintaining stable flow resolution [14].



**Fig. 2.** Mesh of fluid domain of building block

## 2.3 Governing Equation

In Computational Fluid Dynamics, the governing equations are derived from conservation laws. The three fundamental ones are:

- i. Continuity equation (conservation of mass)
- ii. Momentum equation (conservation of momentum (Navier-Stokes))
- iii. Energy equation (conservation of energy (first law of thermodynamics))

### 2.3.1 Continuity equation (mass conservation)

Eq. (1) embodies the principle that mass is conserved in a fluid flow: the rate of change of density in a small control volume plus the net mass flux out of that volume must be zero. In practical terms, it means that fluid cannot spontaneously appear or disappear what flows in must either accumulate or flow out. In CFD, the continuity equation ensures that the discretized flow field respects this fundamental physical constraint, preventing artificial sources or sinks of mass that would otherwise distort the solution as state in [15].

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho u) \quad (1)$$

where  $\rho$ = fluid density,  $u$ = velocity vector  $t$ = time.

### 2.3.2 Momentum equation (Navier–Stokes)

Eq. (2) represents conservation of momentum, applying Newton's second law to a fluid element. It states that the change in momentum which is left-hand side arises from forces acting on the fluid: pressure gradients push the fluid, viscous stresses resist motion, and body forces like gravity add or remove momentum. In CFD, this equation is central because it governs how velocity fields evolve in response to forces it captures how pressure and friction such as viscosity shape the flow's behaviour, including acceleration, deceleration, and shear.

$$\rho \left( \frac{\partial \rho}{\partial t} + \nabla \cdot (\rho u) \right) = -\nabla p + \nabla \times \tau + \rho b \quad (2)$$

where  $p$ = pressure,  $\tau$ = viscous stress tensor,  $b$ = body force per unit mass.

## 2.4 Boundary Condition

This work is intended to define the numerical setup for this study, the simulations are conducted under steady-state conditions using the Standard k-epsilon turbulence model to accurately capture the atmospheric boundary layer effects around the structures. The inlet boundary conditions are prescribed with uniform velocities of 2 m/s, 5 m/s and 8 m/s to represent typical Malaysian wind profiles for both the single and double building geometries. At the outlet, a pressure-outlet condition is applied with a gauge pressure of 0 Pa to simulate an open environment. Finally, the surfaces of the building blocks are assigned a no-slip wall condition, ensuring that the fluid velocity at the solid boundaries is zero to properly model the development of the boundary layer and subsequent flow separation [16].

### 3. Result

#### 3.1 Grid Independence Test

In order to verify that the simulation results were not affected by the mesh resolution, a grid independence test was conducted. Several meshes with different numbers of nodes were generated, and the pressure drop was recorded show in the Figure 3. The intention of this study to ensure the reliability and accuracy of the numerical results, a grid independence test was conducted by comparing velocity profiles across four different mesh densities 8k, 27k, 52k, and 80k nodes. As shown in the graph, the velocity distribution begins to stabilize as the mesh density increases. While there is a noticeable deviation in the results for the course 8k and 27k meshes, the profiles for the 52k and 80k node configurations appear much more consistent and follow a nearly identical trend, especially in the wake development regions.

In addition, the difference between the 52k and 80k results is minimal, it indicates that the solution has reached a stage of grid independence where further refinement would likely yield diminishing returns in accuracy at the cost of significantly higher computational time. Therefore, the 80k node mesh was selected for all subsequent simulations in this study, as it provides the most refined and stable data for capturing complex flow interactions between the building blocks

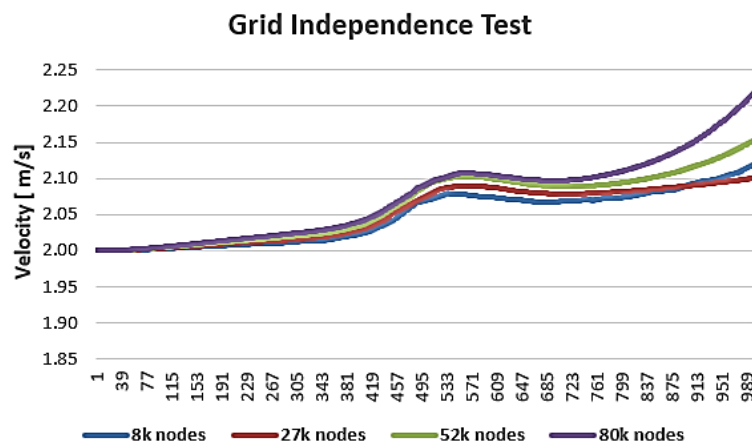
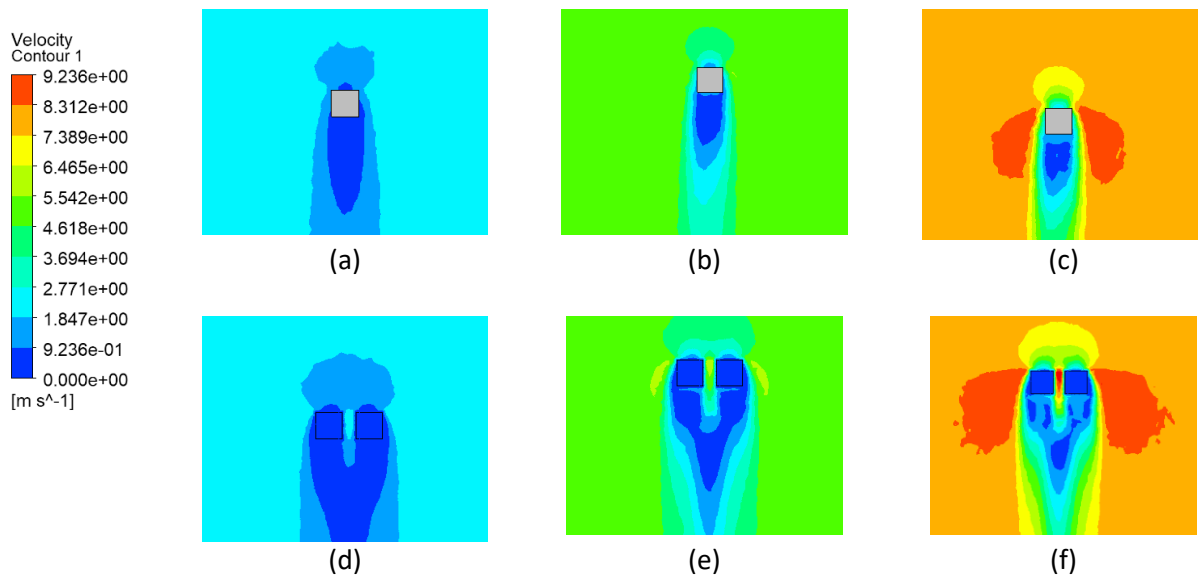


Fig. 3. Grid independence test

#### 3.2 Velocity Contour of Building Block

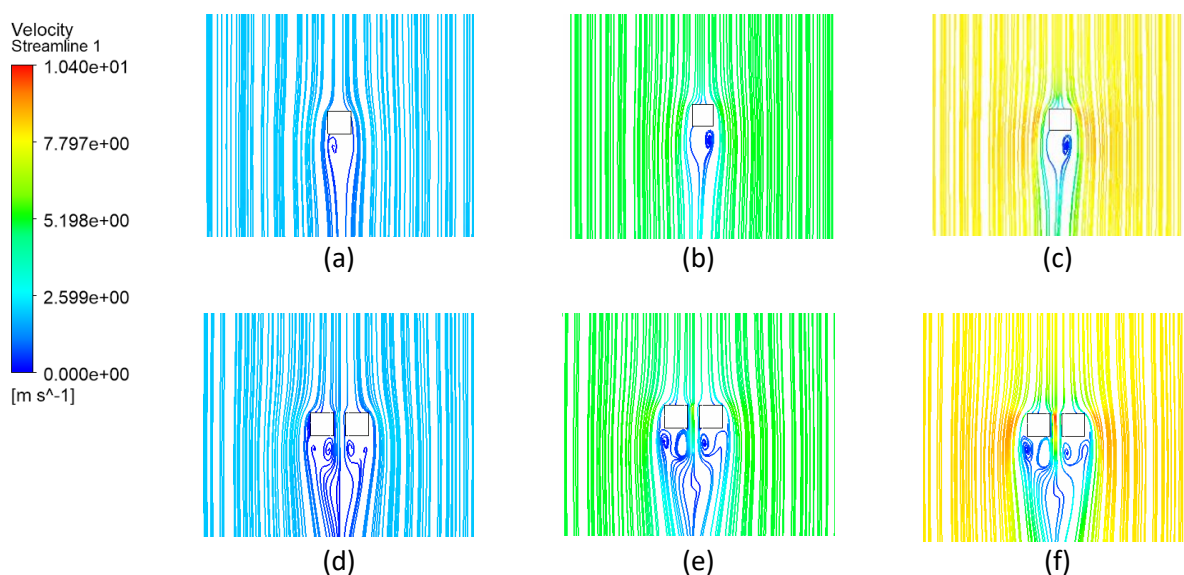
Based on the simulation results shown in Figure 4, the velocity contours clearly illustrate the impact of both inlet velocity and building configuration on the surrounding airflow patterns. In the single block cases shown in Figures 4(a), 4(b), and 4(c), a distinct wake region forms behind the structure, with higher inlet velocities leading to more pronounced flow separation and larger high-velocity zones around the sides of the block. When observing the double block configuration shown in Figures 4(d), 4(e), and 4(f), a significant channelling effect is visible in the gap between the buildings, where the airflow accelerates due to the narrowed passage. Furthermore, the interaction between the two structures creates a more complex and merged wake region compared to the isolated block, demonstrating how neighbouring buildings can substantially alter local wind environments and velocity distributions.



**Fig. 4.** Velocity contour (a) Single block (2 m/s) (b) Single block (5 m/s) (c) Single block (8 m/s) (d) Double block (2 m/s) (e) Double block (5 m/s) (f) Double block (8 m/s)

### 3.3 Velocity Streamline of Building Block

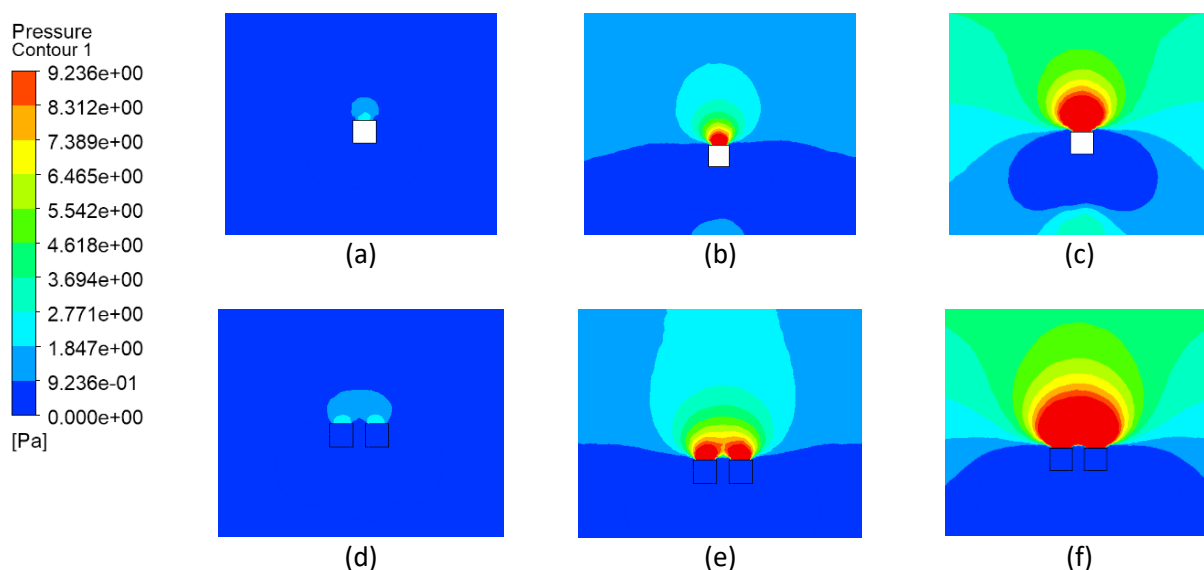
Figure 5 shows the velocity streamline patterns around the single and double building block configurations under different wind velocities. In Figures 5(a) to 5(c), which represent the single building case at 2 m/s, 5 m/s, and 8 m/s respectively, the streamlines indicate smooth flow acceleration around the building edges with a clear recirculation region forming in the wake behind the block, and the wavelength increases as the wind speed increases. For the double building configuration shown in Figures 5(d) to 5(f), stronger flow interaction is observed between the two blocks, where the streamlines become more distorted due to wake interference and flow channelling between the buildings. At higher wind speeds, particularly in Figure 5(f), the wake regions behind both buildings become more pronounced and the velocity magnitude between the blocks increases, indicating stronger aerodynamic interaction compared to the single building case.



**Fig. 5.** Velocity streamline (a) Single block (2 m/s) (b) Single block (5 m/s) (c) Single block (8 m/s) (d) Double block (2 m/s) (e) Double block (5 m/s) (f) Double block (8 m/s)

### 3.4 Pressure Contours of Building Block

Figure 6 illustrates the pressure contour distribution around the single and double building block configurations under different wind velocities. In Figures 6(a) to 6(c), which correspond to the single building case at 2 m/s, 5 m/s, and 8 m/s respectively, a high-pressure region is observed at the windward face of the building due to flow stagnation, while a low-pressure region forms behind the building as the wind speed increases. The magnitude and extent of the high-pressure zone increase with increasing wind velocity, indicating stronger wind loading on the building surface. For the double building configuration shown in Figures 6(d) to 6(f), the pressure distribution becomes more complex due to interaction between the two blocks, with higher pressure accumulation occurring at the front faces and between the buildings. At higher wind speeds, particularly in Figure 6(f), the pressure concentration between the two buildings becomes more pronounced, demonstrating the effect of aerodynamic interference and increased pressure loading compared to the single building case.



**Fig. 6.** Pressure contours (a) Single block (2 m/s) (b) Single block (5 m/s) (c) Single block (8 m/s) (d) Double block (2 m/s) (e) Double block (5 m/s) (f) Double block (8 m/s)

## 4. Conclusions

In conclusion, this study shows that wind behaves very differently when it flows past a single building compared to two buildings side-by-side. While a single block creates a simple wake, the double block setup forces air through the gap, causing it to speed up a channelling effect that becomes much more intense at higher wind speeds like 8 m/s. These results prove that we must consider neighbouring structures and varying wind speeds to truly understand urban airflow in Malaysia. Overall, using CFD allowed us to successfully visualize these complex patterns, providing a cost-effective way to improve building safety and comfort.

## References

- [1] Reda, E., Rozli Zulkifli, and Z. Harun. "Large eddy simulation of wind flow through an urban environment in its full-scale wind tunnel models." *Journal of Mechanical Engineering and Sciences* 11, no. 2 (2017): 2665-2678. <https://doi.org/10.15282/jmes.11.2.2017.9.0243>
- [2] Abd Razak, Azli, Mohd Azhari Mohd Rodzi, Amirul Hakim Jumali, and Sheikh Ahmad Zaki. "Analysis of pedestrian-level wind velocity in four neighbourhoods in Klang Valley." *Jurnal Teknologi (Sciences & Engineering)* 76, no. 5 (2015). <https://doi.org/10.11113/jt.v76.5531>

- [3] Tan, Mohd Hilman Mohd Akil, Mohd Faizal Mohamad, Azli Abd Razak, Nurnida Elmira Othman, and Shahliza Azreen Sarmin. "Numerical Predictions on the Wake Interference Flow in Two-dimensional Street Canyon based on Various RANS Turbulence Closure Models." *Journal of Applied Engineering Design and Simulation* 5, no. 2 (2025): 105-115. <https://doi.org/10.24191/jaeds.v5i2.140>
- [4] Agrawal, Shreya, Jack K. Wong, Jie Song, Oya Mercan, and Paul J. Kushner. "Assessment of the aerodynamic performance of unconventional building shapes using 3D steady RANS with SST k- $\omega$  turbulence model." *Journal of Wind Engineering and Industrial Aerodynamics* 225 (2022): 104988. <https://doi.org/10.1016/j.jweia.2022.104988>
- [5] Swarno, Husna Aini, Sheikh Ahmad Zaki, Aya Hagishima, and Yusri Yusup. "Characteristics of wind speed during rainfall event in the tropical urban city." *Urban Climate* 32 (2020): 100620. <https://doi.org/10.1016/j.uclim.2020.100620>
- [6] Chu, Ruoping, and Kai Wang. "Cfd in urban wind resource assessments: A review." *Energies* 18, no. 10 (2025): 2626. <https://doi.org/10.3390/en18102626>
- [7] Hu, Yongyu, Fusuo Xu, and Zhi Gao. "A comparative study of the simulation accuracy and efficiency for the urban wind environment based on CFD plug-ins integrated into architectural design platforms." *Buildings* 12, no. 9 (2022): 1487. <https://doi.org/10.3390/buildings12091487>
- [8] Xiong, Minghui, Bing Chen, Hua Zhang, and Yao Qian. "Study on accuracy of cfd simulations of wind environment around high-rise buildings: a comparative study of k- $\epsilon$  turbulence models based on polyhedral meshes and wind tunnel experiments." *Applied Sciences* 12, no. 14 (2022): 7105. <https://doi.org/10.3390/app12147105>
- [9] Kabošová, Lenka, Stanislav Kmeť, and Dušan Katunský. "Wind flow around buildings of basic shapes with and without a wind-adaptive envelope." *Selected Scientific Papers-Journal of Civil Engineering* 15, no. 1 (2020): 59-75. <https://doi.org/10.1515/sspjce-2020-0007>
- [10] Pavlidis, Chariton L., Anargyros V. Palampigik, Konstantinos Vasilopoulos, Ioannis C. Lekakis, and Ioannis E. Sarris. "Air flow study around isolated cubical building in the city of Athens under various climate conditions." *Applied Sciences* 12, no. 7 (2022): 3410. <https://doi.org/10.3390/app12073410>
- [11] Mittal, A. K., D. Ghosh, S. Behera, I. A. Siddiqui, and D. S. Dharmshaktu. "Wind flow simulation in the vicinity of tall buildings through CFD." In *Proceedings of The Eighth Asia-Pacific Conference on Wind Engineering, Chennai, India*, pp. 682-690. 2013. [https://doi.org/10.3850/978-981-07-8012-8\\_329](https://doi.org/10.3850/978-981-07-8012-8_329)
- [12] Liu, Sumei, Wuxuan Pan, Xingwang Zhao, Hao Zhang, Xionglei Cheng, Zhengwei Long, and Qingyan Chen. "Influence of surrounding buildings on wind flow around a building predicted by CFD simulations." *Building and Environment* 140 (2018): 1-10. <https://doi.org/10.1016/j.buildenv.2018.05.011>
- [13] Blocken, Bert, and Ted Stathopoulos. "CFD simulation of pedestrian-level wind conditions around buildings: Past achievements and prospects." *Journal of Wind Engineering and Industrial Aerodynamics* 121 (2013): 138-145. <https://doi.org/10.1016/j.jweia.2013.08.008>
- [14] 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.2014.12.002>
- [15] Ji, Guozhao, Meng Zhang, Yongming Lu, and Jingliang Dong. "The basic theory of CFD governing equations and the numerical solution methods for reactive flows. In: Computational Fluid Dynamics - Recent Advances, New Perspectives and Applications Recent Advances, New Perspectives and Applications." *Intechopen*, London, pp. 1-30. (2023): 1-30. <https://doi.org/10.5772/intechopen.113253>
- [16] Hnaïen, Nidhal, Walid Hassen, Lioua Kolsi, Abdelhakim Mesloub, Mohammed A. Alghaseb, Khaled Elkhayat, and Mohamed Hssan Hassan Abdelhafez. "CFD analysis of wind distribution around buildings in low-density urban community." *Mathematics* 10, no. 7 (2022): 1118. <https://doi.org/10.3390/math10071118>