© December 2024 | IJIRT | Volume 11 Issue 7 | ISSN: 2349-6002

Computational Fluid Dynamics (CFD) Modelling: Designing Building for Wind Oriented Architecture

S Amit Rao[1], Dr. Parampreet Kaur[2]
[1]Undergraduate Student, Amity University Chhattisgarh
[2]Head of Institution, Amity University Chhattisgarh

Abstract*—The integration of Computational Fluid Dynamics (CFD) modeling into architectural design has profoundly transformed the approach to creating wind-oriented buildings. This dissertation investigates the application of CFD techniques to optimize architectural designs for enhanced wind flow management, energy efficiency, and structural integrity. By simulating wind behavior around architectural structures, designers can predict and mitigate potential issues related to wind loads, ventilation, and thermal comfort. The study explores various CFD methodologies, including turbulence modeling and boundary conditions, to accurately represent real-world wind interactions. Through detailed case studies of existing wind-oriented buildings, the research demonstrates the practical benefits and challenges of CFD implementation. The findings emphasize the significance of CFD in achieving sustainable and resilient architectural designs that harmonize with their environmental context, ultimately contributing to the advancement of eco-friendly and energy-efficient building practices.

***Index Terms*— Computational Fluid Dynamics (CFD) Simulation, Wind-Oriented Architecture, Vortex Formation, Natural Ventilation**

I. INTRODUCTION

The increasing emphasis on sustainability and energy efficiency in modern architecture has propelled the exploration of innovative design approaches. One such approach is the use of Computational Fluid Dynamics (CFD) modeling to optimize building designs for wind-oriented architecture. This dissertation aims is to achieve and approach the optimized design strategies for enhanced natural ventilation and its role in energy efficiency of the buildings using CFD simulation in Indian context, thereby reducing reliance on artificial ventilation systems and promoting naturally ventilated systems.

CFD modeling offers a sophisticated method for simulating and analyzing airflow around and within buildings. By utilizing this technology, architects can predict and manipulate wind behavior to achieve desirable outcomes, such as improved ventilation, reduced wind loads. This dissertation will explore the theoretical underpinnings of CFD, its practical applications in architectural design, and the benefits of integrating wind-oriented principles into Architecture.

II. LITERATURE STUDY

A. NATURAL VENTILATION

Natural ventilation is an energy efficient alternative for reducing the energy use in buildings, achieving thermal comfort, and maintaining a healthy indoor environment. Typically, the energy cost of a naturally ventilated building is 40% less than that of an air-conditioned building. Natural ventilation, therefore, contributes to a sustainable environment by reducing energy use in buildings. Natural ventilation has become a new trend in building design in architectural community and has been used in many types of buildings, even in highly indoor climate-controlled hospitals (Camille Allocca, 2003).

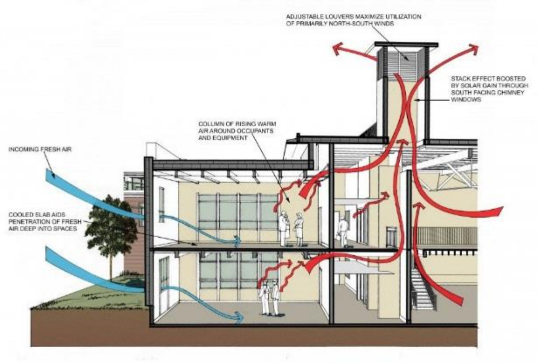

Figure 1- Representation of Natural ventilation followed by stack effect in a building

B. COMPUTATIONAL FLUID DYNAMICS (CFD) MODELLING

Computational Fluid Dynamics (CFD) modeling plays a crucial role in wind-oriented architecture by simulating and analyzing airflow patterns around buildings and structures (Kastner, 2021). This allows architects and engineers to optimize building designs for natural ventilation, energy efficiency, and occupant comfort (Bitsuamlak, 2010). By using CFD, designers can predict how wind will interact with a building, identify potential issues like wind pressure zones or areas of poor ventilation, and make informed decisions to enhance the building's performance (Kastner, 2021).

C. VORTEX FORMATION

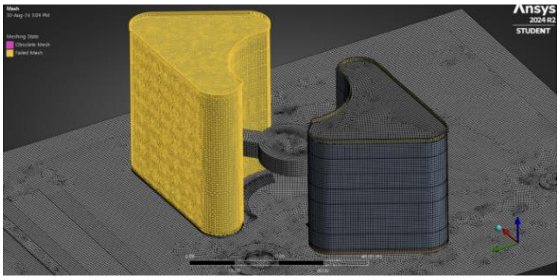

Figure 2- ANSYS CFD Modelling interface

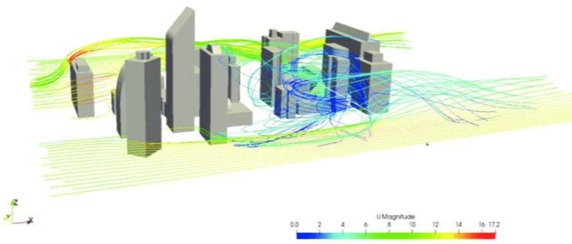

Figure 3- CFD Streamline Analysis
Source -https://www.researchgate.net/publication

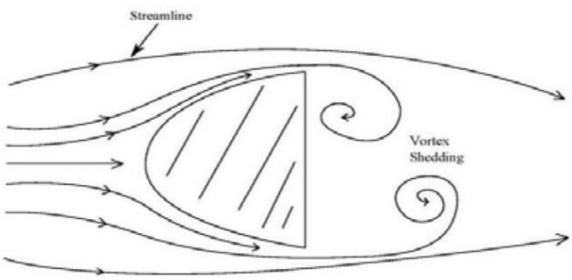

Figure 4- Vortex formation of Burj Al Arab using CFD Analysis
Source - www.cfdsupport.com

Computational Fluid Dynamics (CFD) modeling of vortex formation in architecture is a powerful tool for optimizing building designs to enhance airflow and energy efficiency. By simulating how air moves around and through structures, architects and engineers can identify and mitigate issues like wind pressure zones, turbulence, and poor ventilation. This process helps in designing buildings that naturally regulate temperature, reduce energy consumption, and improve occupant comfort. For example, CFD can be used to study the effects of vortex shedding on tall buildings, ensuring structural stability and reducing wind.

D. STREAMLINE ANALYSIS

Streamline visualization in Computational Fluid Dynamics (CFD) is a high-quality technique for visualization that helps to understand and enhance airflow patterns around and inside buildings in architectural design. Through streamlining, which are the lines that show fluid movement at each point, the architects and the engineers can see how the air moves in the spaces, find out the problems like stagnating zones or very turbulent areas, and thus, make the design decisions to better ventilation and thermal comfort. This method makes the bioclimatic patchwork of living and working spaces more comfortable and efficient by assuring that air circulation is done easily, and thus, the need for mechanical cooling or heating systems is minimized.

E. DESIGN CONSIDERATIONS

When developing buildings for wind-oriented architecture, Computational Fluid Dynamics (CFD) modeling is required to optimize airflow, energy efficiency, and occupant comfort. Key factors include site study to identify prevailing winds and building orientation to maximize natural ventilation. The building's design and aerodynamics should be optimized to reduce drag and turbulence, and ventilation solutions such as cross-ventilation and stack ventilation should be used to increase airflow. The facade design should include sensitive components that adjust to wind conditions, while the inside architecture should facilitate optimal air circulation. Incorporating renewable energy sources and passive design concepts improves overall sustainability. CFD modeling helps to anticipate wind loads and reduce turbulence, maintaining structural integrity and a comfortable atmosphere.

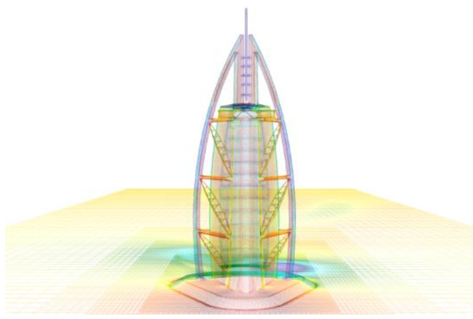

Figure 5- Vortex formation of Burj Al Arab using CFD Analysis
Source - www.cfdsupport.com

III. CASE STUDY

A. RUNWAL BLISS

Owner: Runwal Group

Completion: February 2017

Location: Kanjurmarg East, Mumbai, India

Building Type: Residential Apartments

Total Built-up Area: Approximately 1950 sq. ft. per apartment

Building Name: Runwal Bliss

Vicinity Type: Prime location with excellent rail, road, and proposed Metro connectivity. Close to landmarks like iThink Techno Campus, Eastern

Figure 6- Bird-eye view of the residential apartment Runwal Bliss
Source- https://runwalgroup.in/resid

Nestled in the thriving position of Kanjurmarg East, Mumbai, Runwal Bliss is an exclusive domestic design developed by the recognized Runwal Group. Spread across an emotional breadth, the development offers a mix of luxury and tranquility, making it a sought- after destination for civic residers. Each apartment in Runwal Bliss is courteously designed with commodious layouts and high-end homestretches, icing a comfortable and sophisticated living experience. The design's high position provides excellent connectivity to major corridor of the megacity, along with propinquity to crucial milestones, making it an ideal choice for those seeking a harmonious mix of convenience and fineness in Mumbai. Two conventional approaches i.e. Building energy balance model (BES) and Zonal tailwind network (AFN) models are sluggishly losing their significance and now the focus is more towards CFD grounded simulation which is grounded on mass, instigation and energy conservation in the inflow sphere.

$$\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y}+\frac{\partial w}{\partial z}=0 \quad \text{... (1)}$$

X- Momentum Equation,

$$\rho\left(u\frac{\partial u}{\partial x}+v\frac{\partial u}{\partial y}+w\frac{\partial u}{\partial z}\right)=-\frac{\partial p}{\partial x}+\mu\left(\frac{\partial^2 u}{\partial x^2}+\frac{\partial^2 u}{\partial y^2}+\frac{\partial^2 u}{\partial z^2}\right) \quad \text{... (2)}$$

Y- Momentum Equation,

$$\rho\left(u\frac{\partial v}{\partial x}+v\frac{\partial v}{\partial y}+w\frac{\partial v}{\partial z}\right)=-\frac{\partial p}{\partial y}+\mu\left(\frac{\partial^2 v}{\partial x^2}+\frac{\partial^2 v}{\partial y^2}+\frac{\partial^2 v}{\partial z^2}\right) \quad \text{... (3)}$$

Z- Momentum Equation,

$$\rho\left(u\frac{\partial w}{\partial x}+v\frac{\partial w}{\partial y}+w\frac{\partial w}{\partial z}\right)=-\frac{\partial p}{\partial z}+\mu\left(\frac{\partial^2 w}{\partial x^2}+\frac{\partial^2 w}{\partial y^2}+\frac{\partial^2 w}{\partial z^2}\right) \quad \text{... (4)}$$

Turbulence Equation: K-ε Model

Figure 7 provides a detailed view of airflow between the structures. It highlights qualitative results that are valuable for building planners in making crucial decisions regarding exterior building layouts, spacing, and areas prone to stagnant air pockets caused by wind flow. The current vector plot reveals that, aside from a few areas where the structures are very close together, stagnant zones develop, which are undesirable from a planning perspective

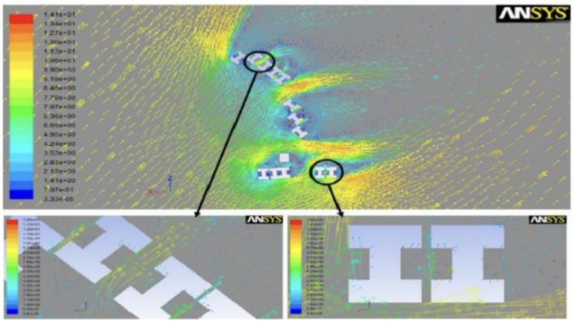

Figure 7- Wind velocity in between two buildings
Source- Author

In conclusion, a three-dimensional numerical model has been developed to analyze the flow patterns around buildings. From the discussions, several key points have emerged. Firstly, Computational Fluid Dynamics (CFD) proves to be an invaluable tool for investigating airflow applications in architecture, offering precise predictions of air velocities around structures. Secondly, the spacing between buildings is crucial and must be thoroughly examined during the planning phase.

B. TAMIL NADU NATIONAL LAW UNIVERSITY (TNNLU)
Owner: Tamil Nadu National Law University (TNNLU)
Completion: 2012
Location: Navalurkuttapattu, Tiruchirappalli, Tamil Nadu, India
Building Type: Educational institution, University campus
Total Built-up Area: Approximately 25 acres (total campus area); the built-up area would be specific to the university buildings

Building Name: Tamil Nadu National Law University

Vicinity Type: Located in a suburban area surrounded by rural landscapes, with close proximity to Tiruchirappalli city

Figure 8- Key plan, Site and selected building for the study
Source- Author

The natural ventilation in terms of wind-driven mode mainly depends on the predominant wind direction and velocity at the location and the performance of the ventilation in apartments can change based on the time, day and season. So annual weather data has been analyzed to know first when natural ventilation mode can be utilized for thermal comfort of occupants. The NBC 2016 states that occupant is comfortable indoors up to 28°C with no or low wind movement. The weather file has been analyzed using honeybee software. In that conditional statement was set to show the wind rose diagram of Tiruchirappalli when dry bulb temperature goes above 28°C. Also, wind velocity beyond 1m/s was considered since lower outdoor velocity won't be helpful for wind driven ventilation indoors.

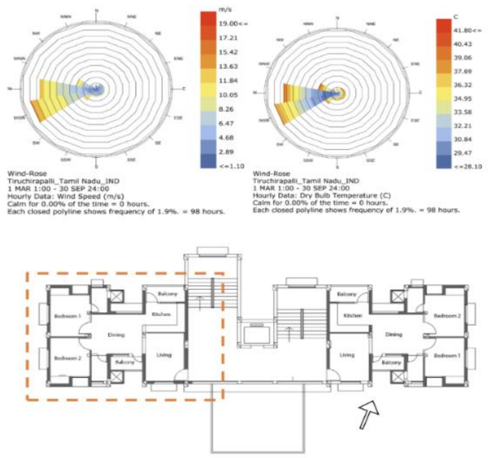

Figure 9- Wind rose analysis with conditions greater than 28- degree Celsius; Building plan

Source- : epw file from ISHRAE and analysis from Grasshopper, honey bee

		North	Sill	South	Sill	West	Sill	Panes
Living	B	-	-	1000*1300	800	-	-	3
	C			1500*1900	200			CW
	D			1500*1300	800			CW
Dining	B	1200*1300	800	1200*1300	800	-	-	3&2
	C	1800*1900	200	600*1300	800			CW
	D	1200*1300	800	600*1300	800			3&2
Kitchen		600*1300	800	-	-	-	-	2
		1200*1300	800					CW
		600*1300	800					2
Bedroom 1		600*1300	800	-	-	1000*1300	800	2&3
		1200*1300	800			1500*1300	800	CW
		600*1300	800			1500*1300	800	2&CW
Bedroom 2		-	-	600*1300	800	1000*1300	800	2&3
				1200*1300	800	1500*1300	800	CW
				1200*1300	800	1200*1300	800	CW

Table 1- Showing the minimum required sizes for the fenestrations as per the results of the simulation

The base case was simulated with wind from predominant wind direction 260° from the north (WEST) and the input of 3.5m/s was given as inlet velocity of air. The windows which were of 3 pane sliding type (1500*1300mm) and 2pane sliding type (1200*1300mm) were considered to be open on the left side as observed by occupant behavior.

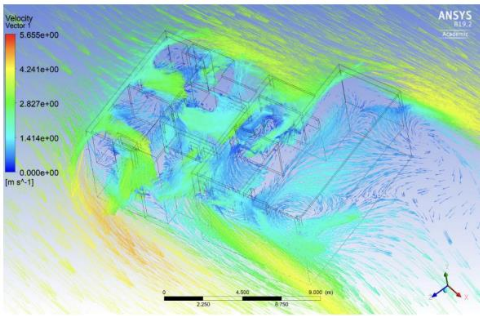

Figure 10 - Streamline Analysis

In Conclusion, The study involves analysis of measurement, simulation of the CFD model of an apartment flat, comparison of them and different configuration for finding better wind distribution and wind velocities. The measurements compared with base case showed a little variation of wind velocity values. The simulation of the base case with predominant wind direction showed that there can be improvements made with the opening arrangement and 2 more cases were simulated to find the best option.

C. CLASSROOM OF IIT BOMBAY

Owner: Indian Institute of Technology Bombay, Ministry of Education, Government of India

Completion: Established in 1958

Location: Powai, Mumbai, Maharashtra, India

Building Type: Educational institution, University campus

Total Built-up Area: Approximately 550 acres (total campus area)

Building Name: Indian Institute of Technology Bombay

Vicinity Type: Urban area, surrounded by residential neighborhoods, Powai Lake, and Sanjay Gandhi National Park

A classroom in the Chemical Engineering Department, IIT Bombay, is used as a test case. The room is modelled as a cuboidal fluid domain containing air. One wall has a door and two air conditioners. The opposite wall has an exhaust near the corner. A second exhaust is tested for different locations at the same height as the first exhaust on three walls. Air enters the room through the air conditioners, circulates inside the room, and leaves through the exhaust fans. The door is maintained at ambient conditions.

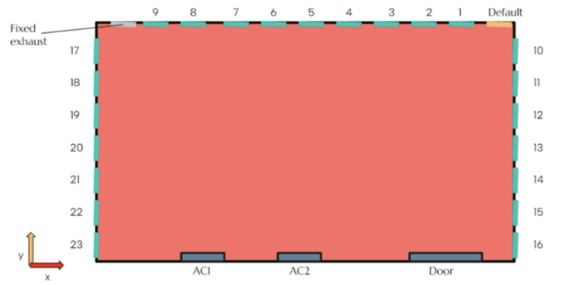

Figure 11- Positions of variable exhaust fans (1-23)
Source- Author

Fig 11 shows the different positions for which the simulation would be run, leading to a total of 24 simulations for the flow field and air residence time. At the end, based on the results, some interventions are introduced in order to minimize the air residence time in specified regions of interest.

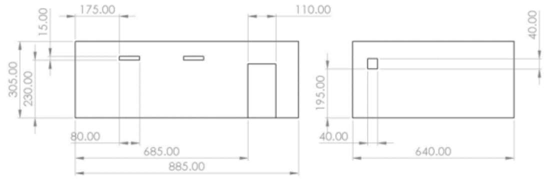

Figure 12- Dimensions of the classroom
Source- Author

The geometry as shown above has the following dimensions. All units are in centimeters. A single door is located near the room's corner and 2 ACs are fixed on the wall which has the door. One of the exhaust fans' positions is fixed on the wall opposite to the door, and the other exhaust fan is moved along the three walls (other than the one containing the ACs) of the room.

In Conclusion, the flow and ventilation analysis of a classroom in Chemical Engineering Department at IIT Bombay was carried out. The air residence time for various placement of exhaust fans was obtained via Open FOAM simulations, and the optimum position of the exhaust fan was determined.

IV. BUILDING PROTOTYPE

The building prototype developed for studying wind analysis through Computational Fluid Dynamics (CFD) simulations serves as a simplified architectural model designed to understand the effects of wind flow around and within building structures. This prototype allows researchers to examine variables such as wind pressure, airflow patterns, ventilation efficiency, and structural impacts across various orientations and configurations.

A. KEY ASPECTS OF THE BUILDING PROTOTYPE

Geometry and Structure: The prototype typically incorporates a simplified geometry, often including a single or multi-story box-like structure with distinct edges, flat or sloped roofs, and openings to mimic windows and doors. For detailed insights, it may feature overhangs, courtyards, or recesses to simulate architectural elements that affect airflow.

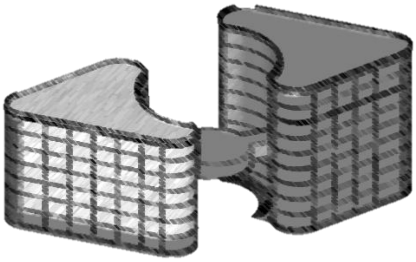

Figure 13- Vortex formation

Material and Surface Properties: To simulate realistic interactions with wind, the prototype is assigned materials with known surface roughness and thermal properties. The choice of material impacts how wind speed and pressure distribute over surfaces, aiding in the study of weathering effects and building envelope performance.

Orientation and Placement: The prototype is rotated or reoriented in CFD simulations to understand the influence of cardinal directions on wind patterns. This helps in optimizing building orientation for natural ventilation, reducing energy consumption, and minimizing wind loads on structural elements.

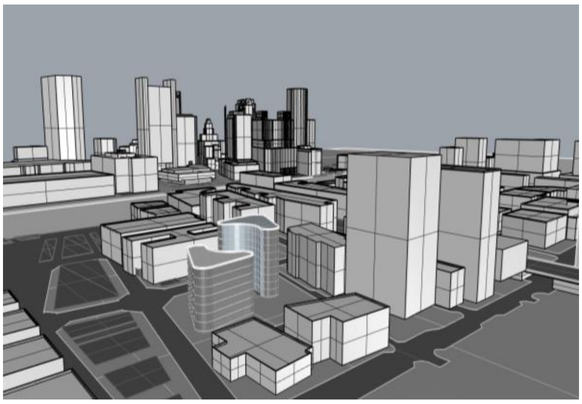

Figure 14- vicinity generation in Rhino Software

Environmental Variables: CFD simulations often incorporate environmental factors like wind speed, turbulence intensity, and atmospheric boundary layer conditions. This allows the prototype to respond dynamically to changes in wind conditions, simulating real-world scenarios and extreme weather events.

Airflow and Ventilation Modeling: Openings in the building prototype are strategically placed to study airflow efficiency within interior spaces. This aspect of CFD analysis provides insights into natural ventilation strategies, air exchange rates, and how interior spaces maintain thermal comfort with minimal mechanical intervention.

Analysis of Results: The simulations generate results on wind pressure distribution, vortex formation, drag forces, and airflow paths. By visualizing these elements, designers can identify zones of high wind load, potential areas of discomfort due to high-speed winds, and spaces where ventilation can be improved.

V. CONCLUSION

This dissertation has established the pivotal role of Computational Fluid Dynamics (CFD) modeling in the architectural design of wind-oriented buildings. By employing CFD, architects and engineers can accurately simulate and analyze wind behavior around structures, providing precise predictions of airflow patterns, wind loads, and thermal comfort levels. This method not only enhances structural integrity and energy efficiency but also aligns with sustainable architectural practices.

The research highlighted the utility of CFD in tackling complex environmental challenges and optimizing building performance. Through case studies and simulations, the importance of integrating wind-oriented principles in the early design phases was emphasized to mitigate issues related to ventilation, air quality, and thermal comfort. CFD modeling emerges as an invaluable tool for achieving resilient, eco-friendly architectural solutions that harmonize with the natural environment.

In summary, integrating CFD modeling into architectural design represents a significant advancement, fostering innovation and sustainability. This dissertation offers a comprehensive framework for future research and practical applications, encouraging the adoption of CFD techniques in design processes. Ultimately, the study underscores the potential of CFD modeling to revolutionize building design, ensuring that structures are aesthetically pleasing, environmentally responsible, and conducive to occupant well-being.

VI. ACKNOWLEDGEMENT

I extend my heartfelt gratitude to my research guide, Dr.Parampreet Kuar for their invaluable guidance and support throughout this research. My sincere thanks to the faculty and staff of Amity University Chhattisgarh for their assistance and resources. Appreciation goes to my colleagues and peers for their encouragement and constructive feedback. Special thanks to the professionals and experts who shared their knowledge and insights. Lastly, I owe my deepest thanks to my family and friends for their unwavering support and patience throughout this journey. This accomplishment is a testament to their belief in me.

VII. REFERNCES

[1] Buildings - A Review. AIVC, 220.
Camille Allocca, Q. C. (2003). Design analysis of single-sided natural ventilation . Semantic Scholar, 18.

[2] Kastner, T. D. (2021). Streamlined CFD simulation framework to generate wind-pressure coefficients on building facades for airflow network simulations. SpringerLink, 1200.

[3] Kim, D. (2013). The Application Of Cfd To Building Analysis And Design: A Combined Approach Of An Immersive Case Study And Wind Tunnel Testing. vtechworks, 254. Retrieved from https://vtechworks.lib.vt.edu/server/api/core/bitstreams/2098bfc9-e182-444f-80d9-9b91b8fdfade/content

[4] Li, B. L. (2015). Revisiting the ‘Venturi effect’ in passage ventilation between two non-parallel buildings. Semantic Scholar , 31.

[5] Lin, L. G. (2006). Sustainable Urban Housing in China: Principles and Case Studies for Low-Energy Design. China: SpringerLink. Retrieved from https://link.springer.com/book/10.1007/978-1-4020-4786-2

[6] Mei, M. A. (2019). A novel methodology for architectural wind environment study by integrating CFD simulation, multiple parametric tools and evaluation criteria. China: SpringerLink. Retrieved from https://link.springer.com/article/10.1007/s12273-019-0591-8