Contents lists available at http://qu.edu.iq

Al-Qadisiyah Journal for Engineering Sciences

Journal homepage: http://qu.edu.iq/journaleng/index.php/JQES

Analysis of aerodynamics around tall buildings with several configurations

Ahmed Ali Shakir Al-Saadi

Department of Mechanical Engineering, College of Engineering, University of AL-Qadisiyah, Ad-Diwaniyah, Iraq.

ARTICLE INFO

Article history:
Received 8 July 2022
Received in revised form 14 August 2022
Accepted 21 September 2022

Keywords:
Aerodynamics
Wind
Tall buildings
CFD
Drag coefficient

ABSTRACT

The streamlined exterior shapes of tall buildings are important to reduce the effect of the wind. Therefore, an examination of different techniques for the exterior design of tall buildings is required. This study aims to analyses some tall buildings to select the most streamlined design in order to reduce high wind risks. The benchmark used in the current study is a building with a height of 120 m and a triangular cross-section with a side length of 20 m. A square cross-section twisted building design is used as a modified model in tall buildings of about 120 m. The rotation angle of the building is 45° for each twisted path. Six configurations of this type of building are tested with different radiuses of fillet on their edges, which are 0, 1, 2, 3, 4, and 5 m respectively. All geometries of the buildings are created by SolidWorks, while mesh and simulations are achieved using ANSYS Fluent. A great agreement is obtained between the current results and the previous related study for the benchmark. Using twisted buildings with a fillet of 5 m can lead to a reduction of the drag coefficient of about 27.5% relative to the benchmark. Wind in a horizontal direction can be reduced by using twisted geometry. But in terms of separation, using a fillet with a large radius can lead to avoiding early separation of air.

1. Introduction

Several criteria determine the design and location of tall buildings in cities. Climate is the most important parameter for designing tall buildings [1-4]. Residential areas' environments should be improved for more comfortable and optimized energy use. And that can lead to a controlled environmental situation in these specific areas. Wind force is one of the most effective climatic factors in tall building designs [5]. A difference in temperature can lead to a change in the static pressure and that can lead to an increase in the velocity of wind [6]. Most tall building designs are more interested in studying wind loads than earthquake loads [7-10]. Architectural design, structural, and mechanical approaches are the main three approaches to designing tall buildings. The structural approach focuses on earthquake and wind loads for tall buildings because these types of buildings are more vulnerable to the external conditions than low-rise buildings [11]. It is necessary for evaluating wind loads in tall buildings and the uncommon exterior designs, especially in buildings with more than 40 stories, because of the changing wind loads fast and suddenly [12]. All architectural designers aspire to minimize the effects of wind on buildings in parallel to reduce the swirls behind the buildings, at corners, and near the ground.

* *Corresponding author.*
E-mail address: *ahmed.shakir@qu.edu.iq* (*Ahmed Al-Saadi*)

https://doi.org/10.30772/qjes.v15i4.855

Nomenclature			
2-D	Two-dimensional	L	Height of the building
3-D	Three-dimensional	r	Fillet radius
CFD	Computational Fluid Dynamics	V	Velocity
A	Frontal area of the building		
C_D	Drag coefficient	Greek symbols	
F_D	Drag force	ρ	Density

Many studies have been conducted for the exterior design optimization of tall buildings. Davenport [13] examined the effect of the exterior design of buildings on the overall behaviour of aerodynamics by using many model tests. Baghaei [14] investigated aerodynamic optimization of high-rise buildings. In his study, he focused on climate-roofing in Iran. 40 prototypes with different climates were used to achieve the optimal design. The optimal design of a building should have a minimum of eddies around the building. In the 1990's, high-rise buildings increased overall in the world, and that led to an increase in the studies of aerodynamics of tall buildings to reduce the effect of wind on this type of building [15-19]. Some researchers examined tapered and set-back design techniques for high-rise buildings [20, 21]. Other techniques of exterior building design, such as openings and spoilers, were used to reduce the effect of the wind on buildings [22, 23]. Kelly et al. [24] studied the influence of helical or twisted building models on the aerodynamics around buildings to resist wind loads. Tse et al. [25] analysed available space and cost regarding tall building designs and their effects on economic terms. Tanaka et al. [26] investigated many cross-sectional models of tall buildings, such as square, rectangular, circular, elliptical, and angle-modified models, in addition to helical and compound models. Some researchers mentioned that changing the exterior design of buildings has been restricted by many design factors, not just aerodynamics [27-29]. Furthermore, different perspectives have been taken during the design of tall buildings, in addition to aerodynamics [30-37]. Pressure coefficients for high-rise buildings have been studied to provide a clear picture of pressure differences [38-39]. while some studies concentrated on the impact of wind on pedestrians [40-43, 56]. Some previous studies used Autodesk Flow Design for simulating the aerodynamics around buildings [2, 44, 45]. Some recent research has focused on the applications of CFD in tall buildings [54, 55].

The main aim of this study is to analyse the drag force on tall buildings. Six different designs of tall buildings with a twist, in addition to the benchmark model, are used to achieve the optimal design for these proposed designs. The building with the minimum drag force is the best in terms of wind aerodynamics. That can lead to a decreased wind load on the building. Studying the aerodynamics of tall buildings is limited in many countries, such as Iraq, and that is the greatest motivation to carry out this numerical study.

2. Numerical approach

The analysis of aerodynamics around high-rise buildings in this research was achieved by using CFD simulation. SolidWorks and ANSYS Fluent were used to create and simulate all configurations of tall buildings. All the 3-D full-scale configurations used in the current work were created using SolidWorks Software because of the complex shape of the twist. ANSYS Fluent was used to create a mesh around the main geometry and simulate all models inside the computational domain.

2.1. Numerical models

Six twisted building models, in addition to the benchmark, were created. The benchmark had a triangular cross-sectional area with sharp edges, as in Daemei et al.'s study [2].

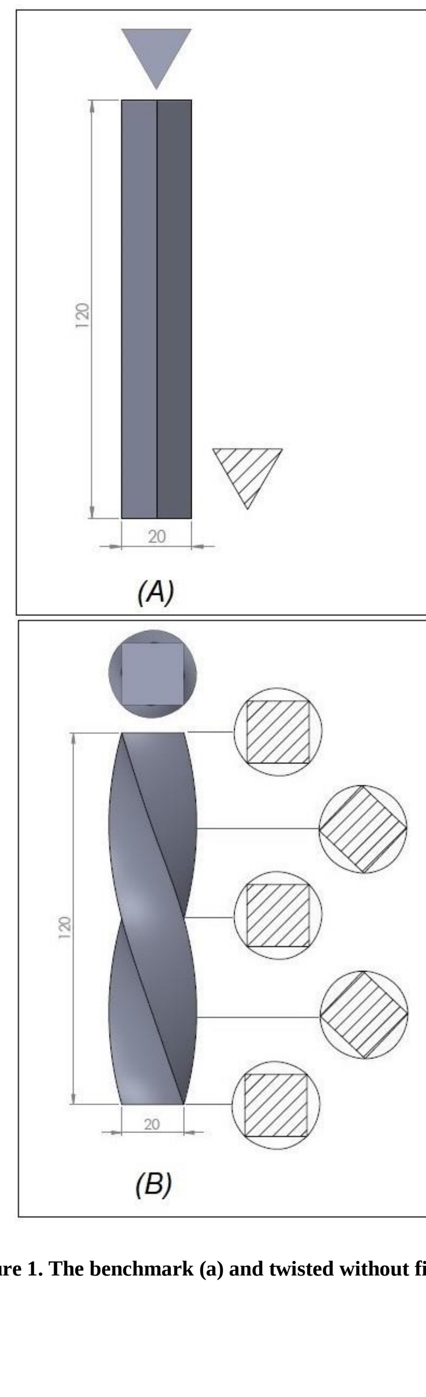

Figure 1. The benchmark (a) and twisted without fillet model (b).

While twisted building models have the same conditions except for the radius of fillet, which was nil, 1, 2, 3, 4, and 5 m respectively. All these buildings have the same height, which was 120 m, and the ratio of height to width was 6. Figure 1 shows the benchmark (a) and twisted without fillet model (b). The overall width of the benchmark model was 20 m, exactly the same as the aspect ratio (L/W=6). It was noted that the width of the cross-sectional area of the twisted model was 20 m. But the overall width of the twisted model was more than 20 m because of the twisted angle.

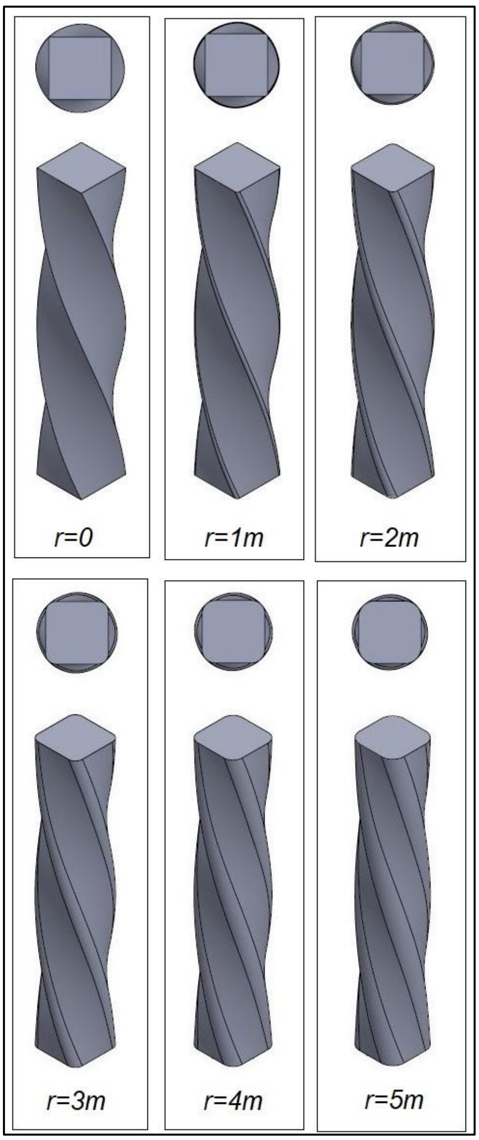

Figure 2. All twisted building models.

Figure 2 illustrates all twisted models in isometric and top views. The first model on the left side is without a fillet. Then the rest, with a radius of 1 m to 5 m, respectively. It was noted that the frontal area of these twisted models was not equal because of the magnitude of the fillet radius. The frontal area of all building models was calculated by using SolidWorks software to achieve more accurate results.

2.2. Computational domain set-up

ANSYS Fluent software was used to create a computational domain around the 3-D geometry of the building. Different dimensions for the computational domain have been used depending on the scale of the model and conditions [46-48]. Figure 3 shows the benckmark model inside the computational domain. Recommended dimensions by Bairagi and Dalui [48] were used in the current study. The distance from the inlet to the tested model was five times the building's height. While the distance from the tested model to the outlet was 15 times the building's height, the overall width of the computational domain was 10 times the building's height, in addition to the width of the building. The overall height of the computational domain was 6 times the building's height.

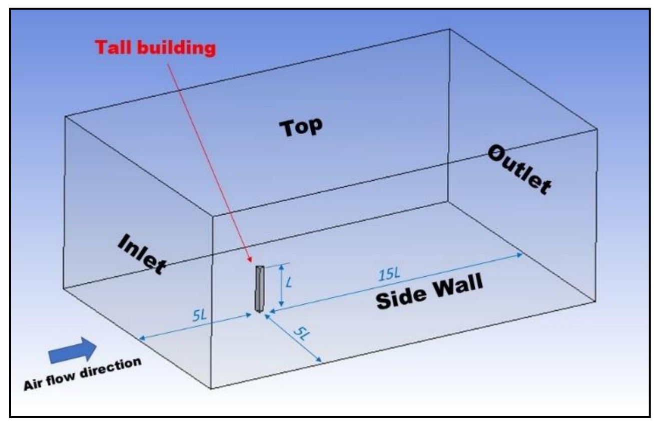

Figure 3. Tall building inside the computational domain

2.3. Mesh refinement

Creating mesh is a very important step to obtaining accurate results. Therefore, mesh refinement, especially near the wall of testing geometry, is crucial. Tetrahedral mesh is used in all cases of the present study because of its sharp edges and twisted geometry. Prismatic cells are added around buildings and over the ground to improve the quality of the mesh. Seven layers of prismatic cells are used as the optimal case in the current study. A growth rate of 1.2 is used in the global domain of mesh. Figure 4 shows the side view of the mesh throughout the computational domain.

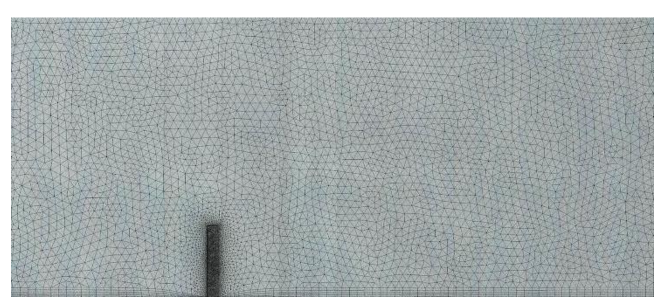

Figure 4. Mesh with seven layers of prismatic cells

Real air movement is not completely horizontal, but it can be in any direction because of the topography of the earth. Some previous studies focused on the direction of air and the effect of it on pedestrian comfort, as shown in Figure 5 [49-53]. The wind speed increases as the distance from the ground increases. The highest velocity of air could occur at a height of about 500 m [14]. In the current study, the direction of air is completely parallel to the horizontal ground of the computational domain.

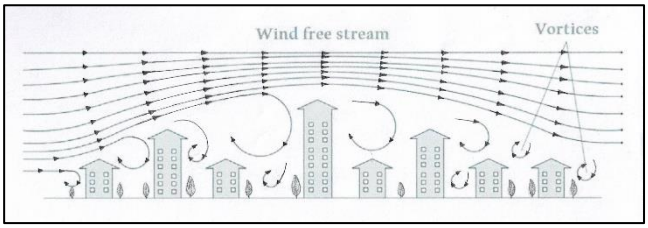

Figure 5. Airflow around buildings [14]

2.4. Drag coefficient

The drag coefficient (C_D) is a dimensionless quantity which represents, in the current study, the resistance of a building against the wind. It is better to decrease this amount as much as possible to increase comfort standards for pedestrians and to decrease serious problems for pedestrians at the same time. Each shape has a different drag coefficient than the others.
This dimensionless quantity can be mathematically clarified by the following equation (1) [2]:

$$C_D = \frac{2\,F_D}{\rho\,A\,V^2} \tag{1}$$

where C_D represents the drag coefficient, and it is a dimensionless quantity. F_D represents the drag force (N). ρ represents the air density (kg/m^3) at 15°C. A represents the frontal area of the building (m^2). V represents wind speed (m/s).

3. Results and discussions

The numerical simulations were achieved in two stages. First, C_D was evaluated for the building with a triangular cross-sectional area as in Daemei et al.'s [2] study to validate the methodology of the present study. Then, six modified exterior shapes of twisted buildings were simulated.

3.1. Validation of numerical results

The mesh dependency test is the major step to achieving accurate numerical results. Therefore, a variety of mesh densities were tested to obtain the optimal number of meshes for this study. Figure 6 shows the mesh dependency test for the benchmark in the present study.
Validation of the numerical simulation results for the benchmark in the current study was accomplished by comparing them with the previous study of Daemei et al. [2] for similar conditions. The benchmark model of the building in the present study had an equilateral triangular cross-sectional area. The length of the side of the triangle was 20 m, while the overall height of this building was 120 m. This baseline model had sharp edges on all sides, exactly similar to the model in the previous study [2]. Figure 7 shows the drag coefficient with the time for the benchmark in Daemei et al. study [2]. They used Autodesk Flow Design to achieve all drag coefficients for many configurations. Figure 8 shows the C_D as a function to iterations for the benchmark in the present study; ANSYS Fluent was applied to obtain all numerical results. Table 1 shows the C_D in the current study and previous study. A good agreement on the benchmark data between the previous and present studies was achieved.

Table 1. Drag coefficients for the benchmark.

	A previous study [2]	Current study
Software	Autodesk Flow Design	ANSYS Fluent
C_D	0.79	0.8
Percentage Error (%)	-----	1.2

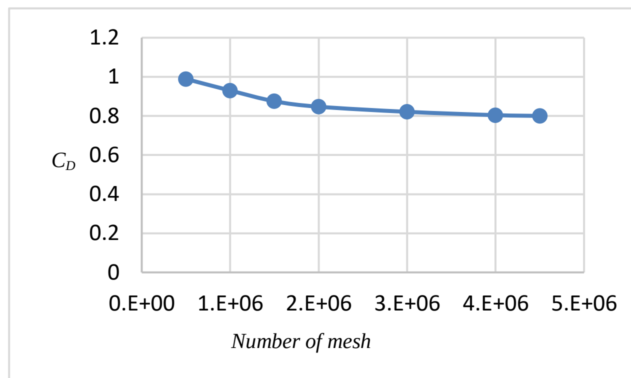

Figure 6. C_D as a function of mesh dependency.

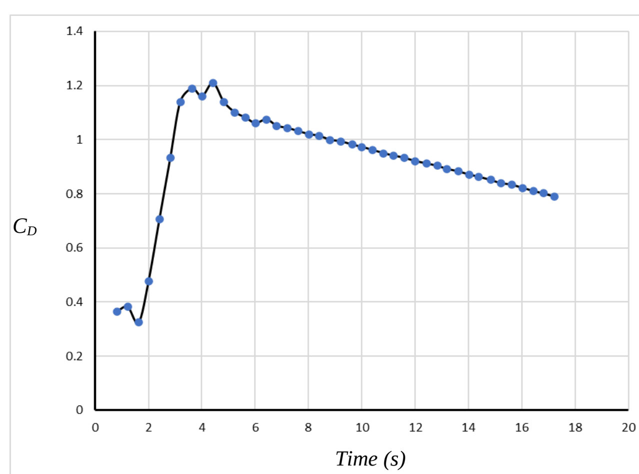

Figure 7. C_D of the basic model in Daemei et al. study [2]

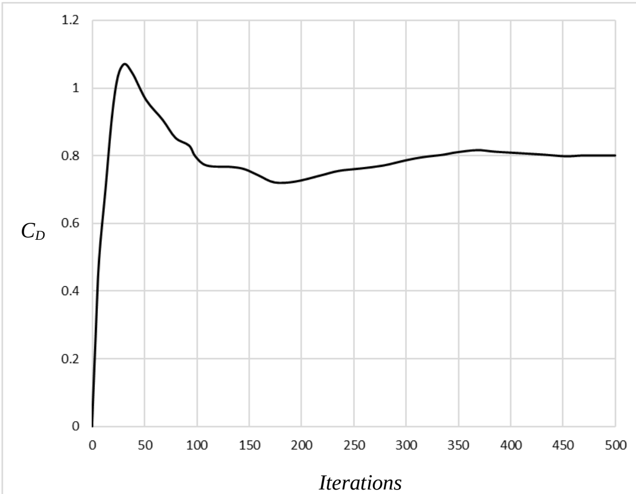

Figure 8. C_D of the basic model in the current study.

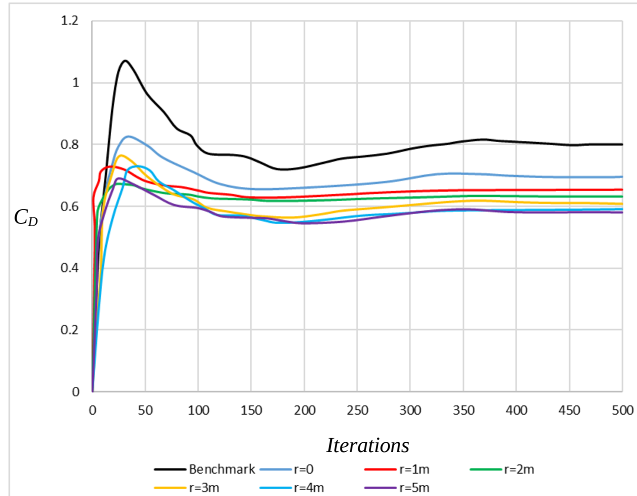

Figure 9. C_D of the modified models and benchmark in the

3.2. Numerical simulation results of modified buildings

The numerical simulations for the modified models of buildings were conducted. The C_D was assessed on twisted models of buildings. Six modified models were simulated. All modified models had the same overall height, cross-sectional shape, and twisted angle. The first model had sharp edges, while the other five had fillet radiuses of 1, 2, 3, 4, and 5 m. The frontal area of these models was not equal because of the different edges. And it was calculated by using SolidWorks for more accurate results. All values of C_D were recorded and evaluated when the airflow was in a stabilization mode. Six modified models, in addition to the benchmark, were investigated. A comparison between all models was employed to figure out which model has the lowest drag coefficient in order to reduce drag force and bending torque. It is clear from Figure 9 and Table 2 that the C_D of the benchmark was about 0.8. While the C_D for all modified buildings was less than 0.7, as shown in Figure 9. The twisted building without a rounded fillet had the highest C_D of the modified group of buildings because of early separation of air. Using fillet on all edges can reduce C_D as well as vortex formations behind the building. According to Figure 9 and Table 2, increasing the radius of the fillet decreases the drag coefficient. Increasing the radius of fillet for edges can lead to decreasing CD, but that can be affected by the capacity of the building. Therefore, a balance between CD and the capacity of the building should be taken into consideration. Twisted tall buildings, on the other hand, are an expensive style to implement, but they are important in earthquake zones, as mentioned in previous studies [2, 24].

Table 2. Drag coefficients for all models

cross-sectional area	Fillet radius	C_D	Percentage reduction (%)
Triangle	r=0	0.8	-----
Square	r=0	0.695	13.125
(twisted)	r=1	0.653	18.375
	r=2	0.631	21.125
	r=3	0.609	23.875
	r=4	0.592	26
	r=5	0.58	27.5

4. Conclusion

Six modified exterior shapes of buildings, in addition to the benchmark, were numerically simulated using ANSYS Fluent to determine the optimal design among them, in proportion to the drag coefficient. According to the numerical simulations of the present work, twisted buildings with a square cross-sectional area and using a radius of fillet of 5 m had the best performance in reducing C_D. Because this technique can result in less swirling wind at the edges. The twist in the building helps the wind flow in an inclined direction. Therefore, the horizontal velocity of air can be decreased to create more comfortable conditions around tall buildings. All results of numerical simulations for modified models were compared to the benchmark and between them as well, in order to define the best exterior shape of a tall building. It is clear that the sixth modification had the lowest C_D, which was 0.58. And that means the C_D was reduced by 27%.

REFERENCES

[1] Mooneghi, M.A. and Kargarmoakhar, R., 2016. Aerodynamic mitigation and shape optimization of buildings. *Journal of building engineering*, *6*, pp.225-235.
[2] Daemei, A.B., Khotbehsara, E.M., Nobarani, E.M. and Bahrami, P., 2019. Study on wind aerodynamic and flow characteristics of triangular-shaped tall buildings and CFD simulation in order to assess drag coefficient. *Ain shams engineering journal*, *10*(3), pp.541-548.
[3] Manzano-Agugliaro, F., Montoya, F.G., Sabio-Ortega, A. and García-Cruz, A., 2015. Review of bioclimatic architecture strategies for achieving thermal comfort. *Renewable and Sustainable Energy Reviews*, *49*, pp.736-755.
[4] Lehmann, S. and Yeang, K., 2010. Meeting with the green urban planner: a conversation between Ken Yeang and Steffen Lehmann on eco-master planning for green cities. *Journal of Green Building*, *5*(1), pp.36-40.
[5] Linden, P.F., 1999. The fluid mechanics of natural ventilation. *Annual review of fluid mechanics*, *31*(1), pp.201-238.
[6] Clancy, L.J., 1975. *Aerodynamics*. John Wiley & Sons.
[7] Irwin, P.A. and Baker, W.F., 2006. The Burj Dubai Tower Wind Engineering. *Structure magazine*, pp.28-31.
[8] Irwin, P., Kilpatrick, J., Robinson, J. and Frisque, A., 2008. Wind and tall buildings: negatives and positives. *The structural design of tall and special buildings*, *17*(5), pp.915-928.
[9] Irwin, P.A., 2009. Wind engineering challenges of the new generation of super-tall buildings. *Journal of Wind Engineering and Industrial Aerodynamics*, *97*(7-8), pp.328-334.
[10] Parker, D. and Wood, A. eds., 2013. *The tall buildings reference book*. Routledge.

[11] Günel, M. and Ilgin, H., 2014. *Tall buildings: structural systems and aerodynamic form*. Routledge.

[12] Moreno, J. ed., 1985. Analysis and Design of High-Rise Concrete Buildings. American Concrete Institute.

[13] Davenport, A.G., 1971. The response of six building shapes to turbulent wind. *Philosophical Transactions of the Royal Society of London. Series A, Mathematical and Physical Sciences*, 269(1199), pp.385-394.

[14] Baghaei, D.A., 2017. *Designing a Model of Affordable Housing in Tall Building with a Focus on the Use of Environmental Factors in Humid Subtropical Climate—Roofing Rasht* (Doctoral dissertation, Department of Architecture, Islamic Azad University).

[15] Kwok, K.C.S., 1988. Effect of building shape on wind-induced response of tall building. *Journal of Wind Engineering and Industrial Aerodynamics*, 28(1-3), pp.381-390.

[16] Isyumov, N., Dutton, R. and Davenport, A.G., 1989. Aerodynamic methods for mitigating wind-induced building motions. In *Structural Design, Analysis and Testing* (pp. 462-470). ASCE.

[17] Dutton, R. and Isyumov, N., 1990. Reduction of tall building motion by aerodynamic treatments. *Journal of Wind Engineering and Industrial Aerodynamics*, *36*, pp.739-747.

[18] Kareem, A., Kijewski, T. and Tamura, Y., 1999. Mitigation of motions of tall buildings with specific examples of recent applications. *Wind and structures*, *2*(3), pp.201-251.

[19] Miyashita, K., Katagiri, J., Nakamura, O., Ohkuma, T., Tamura, Y., Itoh, M. and Mimachi, T., 1993. Wind-induced response of high-rise buildings effects of corner cuts or openings in square buildings. *Journal of Wind Engineering and Industrial Aerodynamics*, *50*, pp.319-328.

[20] Cooper, K.R., Nakayama, M., Sasaki, Y., Fediw, A.A., Resende-Ide, S. and Zan, S.J., 1997. Unsteady aerodynamic force measurements on a super-tall building with a tapered cross section. *Journal of wind engineering and industrial aerodynamics*, *72*, pp.199-212.

[21] Kim, Y. and Kanda, J., 2010. Characteristics of aerodynamic forces and pressures on square plan buildings with height variations. *Journal of Wind Engineering and Industrial Aerodynamics*, *98*(8-9), pp.449-465.

[22] Isyumov, N., Dutton, R. and Davenport, A.G., 1989. Aerodynamic methods for mitigating wind-induced building motions. In *Structural Design, Analysis and Testing* (pp. 462-470). ASCE.

[23] Li, Y., Tian, X., Tee, K.F., Li, Q.S. and Li, Y.G., 2018. Aerodynamic treatments for reduction of wind loads on high-rise buildings. *Journal of Wind Engineering and Industrial Aerodynamics*, *172*, pp.107-115.

[24] Kelly, D., Poon, D., Irwin, P.A. and Xie, J., 2009. Wind Engineering Studies for Shanghai Centre Tower. *RWDI report*.

[25] Tse, K.T., Hitchcock, P.A., Kwok, K.C., Thepmongkorn, S. and Chan, C.M., 2009. Economic perspectives of aerodynamic treatments of square tall buildings. *Journal of Wind Engineering and Industrial Aerodynamics*, *97*(9-10), pp.455-467.

[26] Tanaka, H., Tamura, Y., Ohtake, K., Nakai, M., Kim, Y.C. and Bandi, E.K., 2013. Aerodynamic and flow characteristics of tall buildings with various unconventional configurations. *International Journal of High-Rise Buildings*, *2*(3), pp.213-228.

[27] Li, Y., Zhang, J.W. and Li, Q.S., 2014. Experimental investigation of characteristics of torsional wind loads on rectangular tall buildings. *Structural Engineering and Mechanics*, *49*(1), pp.129-145.

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

[29] Davenport, A.G., 1971. The response of six building shapes to turbulent wind. *Philosophical Transactions of the Royal Society of London. Series A, Mathematical and Physical Sciences*, *269*(1199), pp.385-394.

[30] Tamura, T.E.T.S.U.R.O., Miyagi, T. and Kitagishi, T., 1998. Numerical prediction of unsteady pressures on a square cylinder with various corner shapes. *Journal of Wind Engineering and Industrial Aerodynamics*, *74*, pp.531-542.

[31] Tamura, T. and Miyagi, T., 1999. The effect of turbulence on aerodynamic forces on a square cylinder with various corner shapes. *Journal of Wind Engineering and Industrial Aerodynamics*, *83*(1-3), pp.135-145.

[32] Hu, J.C., Zhou, Y. and Dalton, C., 2006. Effects of the corner radius on the near wake of a square prism. *Experiments in fluids*, *40*(1), pp.106-118.

[33] Mendis, P., Ngo, T., Haritos, N., Hira, A., Samali, B. and Cheung, J., 2007. Wind loading on tall buildings. *Electronic Journal of Structural Engineering*.

[34] Kikitsu, H., Okuda, Y., Ohashi, M. and Kanda, J., 2008. POD analysis of wind velocity field in the wake region behind vibrating three-dimensional square prism. *Journal of Wind Engineering and Industrial Aerodynamics*, *96*(10-11), pp.2093-2103.

[35] Ming, G., Huang, P., Tao, L., Zhou, X. and Fan, Z., 2010. Experimental study on wind loading on a complicated group-tower. *Journal of Fluids and Structures*, *26*(7-8), pp.1142-1154.

[36] Tanaka, H., Tamura, Y., Ohtake, K., Nakai, M. and Kim, Y.C., 2012. Experimental investigation of aerodynamic forces and wind pressures acting on tall buildings with various unconventional configurations. *Journal of Wind Engineering and Industrial Aerodynamics*, *107*, pp.179-191.

[37] Cermak, J.E., 1975. Applications of fluid mechanics to wind engineering—a Freeman Scholar lecture.

[38] Amin, J.A. and Ahuja, A.K., 2014. Characteristics of wind forces and responses of rectangular tall buildings. *International Journal of Advanced Structural Engineering (IJASE)*, *6*(3), pp.1-14.

[39] Kumar, S., YNVM, R. and Kumar, P.R., 2015. Aerodynamic studies of non–circular high–rise buildings. *International Journal of Engineering Research in Mechanical and Civil Engineering*, *12*(5), pp.14-21.

[40] Tominaga, Y., Mochida, A., Yoshie, R., Kataoka, H., Nozu, T., Yoshikawa, M. and Shirasawa, T., 2008. AIJ guidelines for practical applications of CFD to pedestrian wind environment around buildings. *Journal of wind engineering and industrial aerodynamics*, *96*(10-11), pp.1749-1761.

[41] Blocken, B., Carmeliet, J. and Stathopoulos, T., 2007. CFD evaluation of wind speed conditions in passages between parallel buildings—effect of wall-function roughness modifications for the atmospheric boundary layer flow. *Journal of Wind Engineering and Industrial Aerodynamics*, *95*(9-11), pp.941-962.

[42] Mochida, A. and Lun, I.Y., 2008. Prediction of wind environment and thermal comfort at pedestrian level in urban area. *Journal of wind engineering and industrial aerodynamics*, *96*(10-11), pp.1498-1527.

[43] Stathopoulos, T., 2009, July. Wind and comfort. In *5th European and African Conference on Wind Engineering, EACWE 5, Proceedings*.

[44] Flow design preliminary validation brief; 2014. http://download.autodesk.com/us/flow_design/Flow_Design_Preliminary_Validation_Brief_01072014.pdf

[45] Fadl, M.S. and Karadelis, J., 2013. CFD simulation for wind comfort and safety in urban area: A case study of Coventry University central campus. *International journal of architecture, engineering and construction*, *2*(2), pp.131-143.

[46] Elshaer, A., Bitsuamlak, G., El Ansary, A. and Sobhy, M., 2018. Alteration of wind load on tall buildings due to fluid structure interaction. In *In Proceedings of Canadian Society of Civil Engineering Conference (CSCE)* (Vol. 2018, pp. 1-13).

[47] Bairagi, A.K. and Dalui, S.K., 2018, December. Comparison of pressure coefficient between square and setback tall building due to wind load. In *SEC18, Proceedings of the 11th Structural Engineering Convention—2018. Jadavpur University, Kolkata, India*.

[48] Bairagi, A.K. and Dalui, S.K., 2018. Comparison of aerodynamic coefficients of setback tall buildings due to wind load. *Asian Journal of Civil Engineering*, *19*(2), pp.205-221.

[49] Hang, J., Li, Y. and Sandberg, M., 2011. Experimental and numerical studies of flows through and within high-rise building arrays and their link to ventilation strategy. *Journal of Wind Engineering and Industrial Aerodynamics*, *99*(10), pp.1036-1055.

[50] Aly, A.M. and Bresowar, J., 2016. Aerodynamic mitigation of wind-induced uplift forces on low-rise buildings: A comparative study. *Journal of Building Engineering*, *5*, pp.267-276.

[51] Daemei, A.B., Limaki, A.K. and Safari, H., 2016. Opening performance simulation in natural ventilation using design builder (case study: a residential home in Rasht). *Energy Procedia*, *100*, pp.412-422.

[52] Stathopoulos, T. and Blocken, B., 2016. Pedestrian wind environment around tall buildings. In *Advanced Environmental Wind Engineering* (pp. 101-127). Springer, Tokyo.

[53] Stathopoulos, T., Wu, H. and Bédard, C., 1992. Wind environment around buildings: a knowledge-based approach. *Journal of Wind Engineering and Industrial Aerodynamics*, *44*(1-3), pp.2377-2388.

[54] Moosavi, S.H. and Delfani, S., 2020. AIJ Guidelines for Practical Applications of CFD to Pedestrian Wind Environment around Buildings. *Building Engineering & Housing Science*, *13*(3), pp.17-23.

[55] Kataoka, H., Ono, Y. and Enoki, K., 2020. Applications and prospects of CFD for wind engineering fields. Journal of Wind Engineering and Industrial Aerodynamics, 205, p.104310.

[56] Wu, Y., Zhan, Q. and Quan, S.J., 2021. Improving local pedestrian-level wind environment based on probabilistic assessment using Gaussian process regression. *Building and Environment*, *205*, p.108172.