Wind and Structures, *Vol. 29, No. 2 (2019) 139-147*
DOI: https://doi.org/10.12989/was.2019.29.2.139

Study on aerodynamic shape optimization of tall buildings using architectural modifications in order to reduce wake region

Abdollah Baghaei Daemei[1a] and Seyed Rahman Eghbali[*2]

[1]Department of Architecture, Rasht Branch, Islamic Azad University, Rasht, Iran
[2]Department of Architecture, Imam Khomeini International University, Qazvin, Iran

(Received February 26, 2018, Revised May 28, 2019, Accepted May 31, 2019)

Abstract. One of the most important factors in tall buildings design in urban spaces is wind. The present study aims to investigate the aerodynamic behavior in the square and triangular footprint forms through aerodynamic modifications including rounded corners, chamfered corners and recessed corners in order to reduce the length of tall buildings wake region. The method used was similar to wind tunnel numerical simulation conducted on 16 building models through Autodesk Flow Design 2014 software. The findings revealed that in order to design tall 50 story buildings with a height of about 150 meters, the model in triangular footprint with aerodynamic modification of chamfered corner facing wind direction came out to have the best aerodynamic behavior comparing the other models. In comparison to the related reference model (i.e., the triangular footprint with sharp corners and no aerodynamic modification), it could reduce the length of the wake region about 50% in general. Also, the model with square footprint and aerodynamic modification of chamfered corner with the corner facing the wind could present favorable aerodynamic behavior comparing the other models of the same cluster. In comparison to the related reference model (i.e., the square footprint with sharp corners and no aerodynamic modification), it could decrease the wake region up to 30% lengthwise.

Keywords: tall building; shape optimization; wind effects; aerodynamic modification; wake region; wind tunnel simulation

1. Introduction

Aerodynamic behavior is an important characteristic of tall and ductile buildings, so aerodynamic design can play a key role in reducing the wind effect on these buildings (Shiqing *et al*. 2017, Ali and Armstrong 1995, Holmes 2001, Irwin 1995, Irwin 2006, Irwin *et al*. 2008, Kareem *et al*. 1999, Schueller 1977, Scott *et al*. 2005). A tall building's response to wind can be controlled by application of aerodynamic improvements to building's design in order to manipulate the wind flow pattern and break the effective wind force acting on the structure. In practice, such improvement efforts are called "aerodynamic modifications" (Günel and Ilgin 2014, Xie 2014, Elshaer *et al*. 2017, Adamek *et al*. 2017). Aerodynamic modifications are limited to exterior shape modifications and do not alter the general dimensions and structure of the building, so they are typically used when the need for change is purely aerodynamic in nature. Conventional methods of aerodynamic modification include corner modifications such as corner rounding, chamfering, and recessing (Xie 2014, Ilgin Günel 2007, Xie 2012).

The overall shape of a building can play a significant role in mitigating the vortex shedding and aeroelastic effects, so the effect of aerodynamic modifications on a building's wind response could be considerable. Making these modifications in a building's corners could lead to it having a better aerodynamic response than a bluff body building (Xie 2012). It can be argued that desirable flow pattern around the building can be achieved by proper selection of a building's general form and then applying suitable aerodynamic modifications. In this context, many researchers have studied the relation of buildings' aerodynamic and formal features with their response to wind effects (Ilgin and Günel 2007, Amin and Ahujab 2010, Kwok 1988, Tanaka 2013).

The effects of turbulent winds can be mitigated by taking appropriate measures to weaken them particularly with the help of aerodynamic design solutions (Irwin 1995, Chan *et al*. 2009). The goal of architectural aerodynamics is to mitigate the wind forces acting on the building and prevent vortex shedding in the bottom, sides and back of the building (Parker and Wood 2013). Review of the research literature shows that while the aerodynamic design of tall buildings and the effects of drag force, pressure factor, and wind on footways have been extensively researched, the wake region at the back of such buildings has not received the same level of attention. The aim of the present study, was to an investigation of aerodynamic design and modification and assess wake region behind of tall buildings, as this region can significantly affect the design of neighboring structures.

[*]Corresponding author, Ph.D.
E-mail: s.r.eghbali@ikiu.ac.ir
[a] MSc.
E-mail: baghaei@iaurasht.ac.ir

http://www.techno-press.com/journals/was&subpage=7 ISSN: 1226-6116 (Print), 1598-6225 (Online)

2. Research background

Hayashida and Iwasa (1990) conducted a study on the aerodynamic shape of tall buildings and its impact on vortex-induced vibrations of these building. In this study, eight shapes with identical size of 600 m and identical density of 125 kg/m3 were examined. The results of computer simulations conducted in this study provided some information about the aerodynamic behavior and the vortex effect due to the shape of tall building. Dutton and Isyumov (1990) performed a similar study on the effect of aerodynamic treatments on the motion of tall building. The aerodynamic solution evaluated in this study was the use of an opening in the design. Their results showed that the use of an opening can reduce a building' wind induced vibration frequency by about 15%. In a study by Kawai (1998) the effects of corner modifications on aeroelastic instabilities of tall square plan buildings were investigated. The results of this study showed that for an object with square footprint, Rounded corners have a better effect on the wind response than corner cuts and recessed. It was also found that the b/B ratio of 0.05 percent (representing the size of cuts and recessed) is very suitable for the stability of square plan buildings.

Also Tamura and Miyagi (1999) studied the effect of turbulence of aerodynamic forces on a square object with modified corners. This study was concentrated on three corner shapes: Sharp corners, Rounded corners, and Chamfered corners. Their results showed that Rounded and Chamfered corners can reduce the drag force and mitigate the wake region. In a study by Kim and Kanda (2010) they examined the features of aerodynamic forces and pressures acting on square plan buildings with height variations. In this study, buildings were modeled with 5% setback and gradual 10% tapering in height. The results showed that tapering and setback can reduce vibrations and drag force. It was also found that compared to tapering, setback can be up to 40% more effective in reducing the said factors. Kim, Kanda *et al.* (2011) also conducted a study on the effect of wind on the motion of tall buildings with square plan with height variations. In this study, square plan buildings with and without two types of tapering and one type of setback were examined. This study found that the models with tapering and setback have a lower torsional acceleration than the unmodified square models.

The aerodynamic design of tall buildings has been investigated from other viewpoints as well (Cermak 1975, Tamura *et al*. 1998, Hu *et al.* 2006, Kikitsu 2008, Gu 2010, Tanaka *et al.* 2012) The effect of wind pressure factor on tall buildings has also been studied by several researchers (Amin and Ahuja 2014, Yechuri *et al.* 2015). Some researchers have also investigated the impact of wind on sidewalks and pedestrian level (Blocken *et al.* 2007, Tominaga *et al.* 2008, Mochida and Lun 2008, Yasa 2016). Baghaei Daemei (2019) has studied wind effect at the pedestrian level of a tall building. This numerical study carried out through CFD approach using Autodesk Flow Design software, on a real condition. The results show that the trees can reduce wind velocity for human comfort in urban spaces.

Zhi *et al*. (2015) have suggested that direct monitoring of wind force effects on extremely tall buildings is a quite difficult task. They have estimated wind forces on very tall buildings based on limited structural answers. The method of study is through wind tunnel test on an extremely tall building. Gu *et al.* (2014) have conducted a study on the effects of aerodynamic modifications to building cross-section. Including Chamfered, slotted and tapered cross-sections, on the across-wind aerodynamic damping ratio. The results show that the Chamfer ratios can decrease the wind effect on tall buildings. There is also a study carried out by Wahrhaftig and da Silva MA (2018) on high-rise residential building regarding the drag coefficient using computational fluid dynamics on regarding drag the actual building geometry.

Furthermore, Baghaei Daemei *et al*. (2019) have studied the wind aerodynamic and flow characteristics of a triangular-shaped tall building together with a CFD simulation to assess drag coefficient. The results show that aerodynamic modification of Rounded and tapered corners are capable to cause a reduction in the drag coefficient of the building by 66% and 24%, respectively.

3. Material and methods

3.1 Simulation details and analysis methods

The 3D models were constructed using AutoCAD 2014 and then numerical wind tunnel simulations were performed using Autodesk Flow Design 2014 software. STL models were generated and imported into Flow Design. In order to clearly show the flow, the resolution level of the simulation was set at 120%. These simulated wind loads were used to estimate along-wind responses of a tall building, which are less narrow-banded processes, based on the state-space variable approach. To investigate the wind flow pattern, the fluid flow was chosen as plane 2D so that the effect of cross section could be evaluated in relation to the building models. The assessment was carried out on Z coordinate at 2/3 height of the building (Fig. 1).

To evaluate Autodesk Flow Design software, the Autodesk organization has made various simulations and compared to the actual experimental tests. According to their report published in 2015, the focal aspect of this software is on machines and architectural studies. Evaluation in an architectural manner was assessed by a research entitled "Flow Design Preliminary Validation Brief" (2014) comparing to the results of Fadl and Karadelis (2013). For this comparison, an actual building located in Coventry University Central Campus has been simulated in Flow Design and Fluent environments under the same conditions. Outcomes of the research revealed an offset error by 6% from this software to experimental tests and computational simulations by Fluent which makes the software results acceptable. Furthermore, the Flow Design solver uses an LES (Large Eddy Simulation) turbulence model to account for turbulence within the wind tunnel. LES is a mathematical model for turbulence used in computational fluid dynamics of the atmospheric boundary

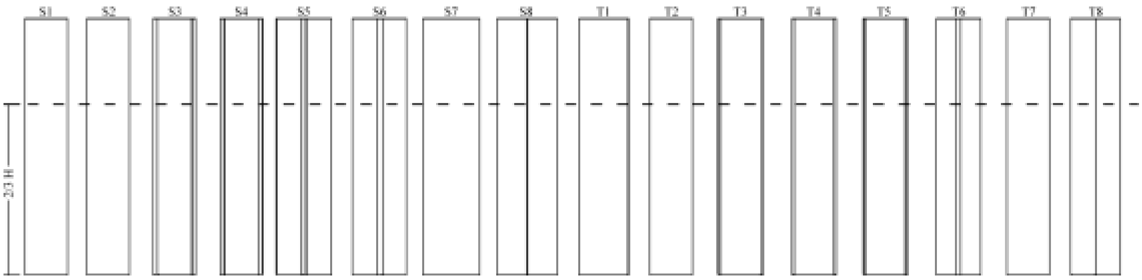


Fig. 1 Characteristics of the point to be assessed on Z coordinate to investigate building wake region

Table 1 Configuration of Flow Design Software

Mesh size (Resolution) (%)	Status	Time (s)		Analysis Type
120	Stabilized	30		2D
Turbulence Model	Wind Tunnel Size (m)			Wind Speed (m/s)
	L	W	H	
LES	1300	300	300	10

layer (Sullivan *et al.* 1994, Zhiyin 2015). Information entered into the software could be extracted from Table 1.

3.2 Outline of wind tunnel simulation models

Tall buildings model of this survey is installed on both square (Tanaka *et al*. 2012) and triangular bases of equal sizes (Tanaka *et al.* 2013, Xie 2014) with three aerodynamic modifications including Chamfered, Recessed and Rounded corners in comparison with the Sharp states as reference. The height of building H=150 m (about 50 story) and the width of 25 m (B). Edit proportions were carried out on Chamfered, Recessed and Rounded models as 1/10 b/B (Günel and Ilgin 2014) (Table 2). Models were located against the wind direction in such a manner that first wind flow was fixed but models were first perpendicular to the flow and next time, the tilted section was placed against the wind flow. In total, 16 states were investigated in such a manner that square models were placed in an angle of 45 degrees and triangular models in an angle of 60 degrees from the center along the wind direction. Models were built with H/B=6 and B/b=1 proportions. Table 2 shows information on models.

4. Design approaches against wind excitation

4.1 Aerodynamic modifications

Aerodynamic optimization can be called “aerodynamic modification”. Aerodynamic architectural modifications consist of corner modifications that do not significantly alter the existing architectural design. Modifications to corner including recessing, chamfering and rounding reduce the across wind building response, as compared with an original building shape with Sharp corners. In a prismatic building, Recessed(notched), Chamfered (cut) and Rounded corners (Fig. 2) can reduce the along-wind and across-wind building response to an important degree (Mochida and Lun 2008, Blocken *et al*. 2007). An altered/modified corner (including Recessed, Chamfered and Rounded corners), which reduces the width of the building by 10 per cent in comparison to a Sharp corner one, reduces the along-wind building response by 40 per cent and the across wind building response by 30 per cent (Kim and Kanda 2010).

Irwin (2009) terms “modified corners” as “softened corners” and states that “The corner softening should extend about 10 per cent of the building width in from the corner (Miyashita *et al*. 1993).” However, corner modifications may cause adverse effects in serviceability and safety of the building. Rounded corners are the most effective type of corner modification. Approximating a circular plan form by increasing the corner Roundness also reduces the wind loads affecting the building to an important degree (Kwon and Kareem 2013). The overall shape of a building can play a significant role in mitigating the vortex shedding and aero elastic effects, so the effect of aerodynamic modifications on a building’s wind response could be considerable. Making these modifications in a building’s corners could lead to it having a better aerodynamic response than a bluff body building (Kim and Kanda 2013).

Optimum design of the tall buildings and use of the aerodynamic modifications can reduce the movement or change the flow pattern around and through buildings. For instance, modifications in the corners of the cross sectional shape of the building, relative to the most frequent strong wind direction are among the minor modification methods that can be utilized to improve the wind performance of tall buildings. These methods can result in reductions in both along wind and across wind responses compared to plain rectangular shape buildings. Hence, the other researchers conducted various studies between aerodynamic modifications and building responses to the wind forces (Xie 2014, Xu and Xie 2015). Earlier records of such studies can be found in the studies by Miyashita *et al.* (1993), Gu and Quan (2004), and Sharma *et al.* (2018).

4.2 Separation and wakes

The wake is the region of disturbed flow (often turbulent) downstream of a solid body moving through a fluid, caused by the flow of the fluid around the body. In Sharp corners body (bluff body) typically produce a separated flow, but they are not the only cause of

Table 2 Configuration of the test models

Square				Triangular			
S1: Sharp	S2: Chamfered	S3: Rounded	S4: Recessed	T1: Sharp	T2: Chamfered	T3: Rounded	T4: Recessed

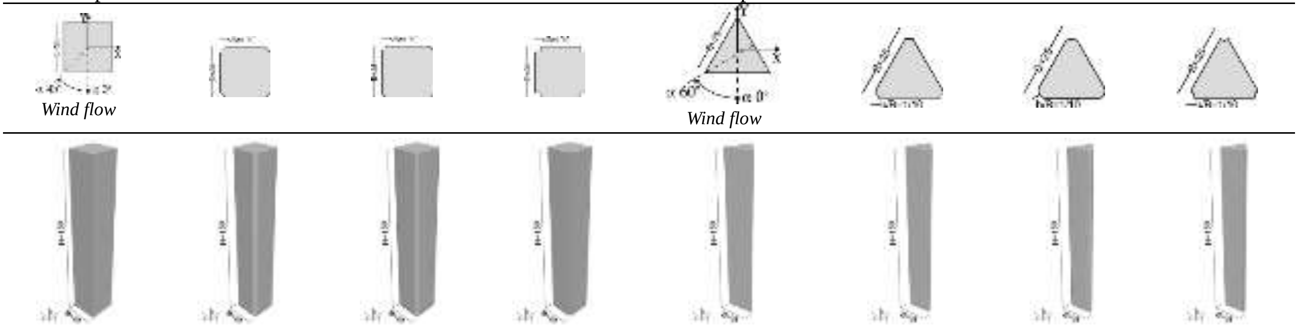


Fig. 2 Aerodynamic modifications

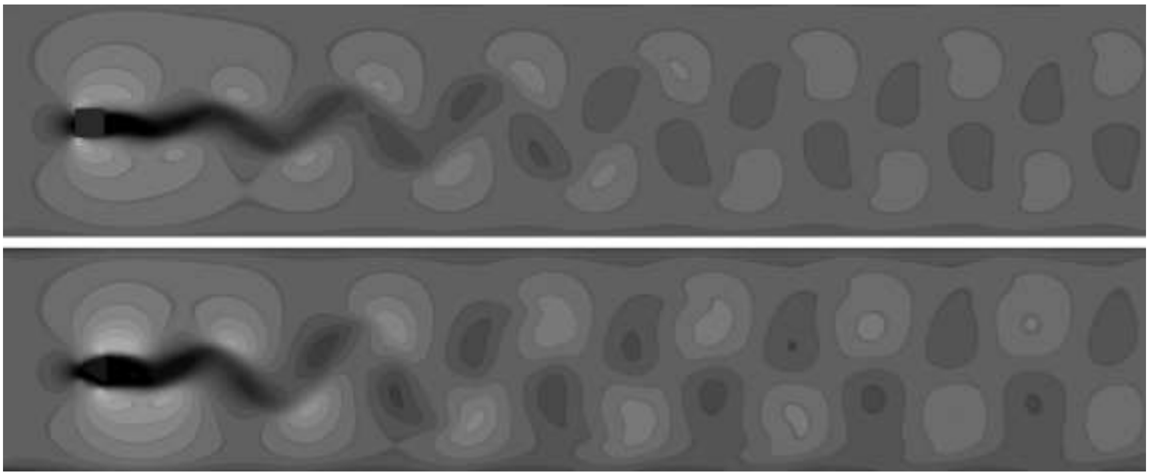


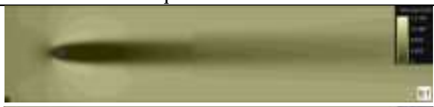
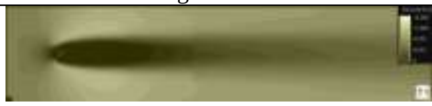
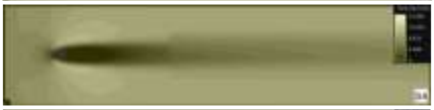
Fig. 3 Flow behind square and triangular shapes

separation. For example, the flow over cylinder can produce a large region of separated flow downstream of the cylinder. This region is called a wake (Fig. 3). The flow patterns over the front and back of the bluff or blunt body shapes are quite different. In the front, the flow smoothly passes over the shapes, but in the wake the flow is usually highly unsteady and large eddies or vortices are shed downstream. The large eddies are formed at a regular frequency and they produce pressure disturbances in the flow which we can sometimes hear as sound waves. Wakes almost always contain large eddying motions which are shed downstream, although they may not shed at a regular frequency, which is called in fluid dynamics "a Kármán vortex street". A vortex street is a repeating pattern of swirling vortices caused by the unsteady separation of flow of a fluid around blunt bodies (Wille 1960).

5. Models evaluation based on the findings

It could be stated that in tall buildings the wake region shall have a significant relationship with building's aspect ratio, shape and their location vis-à-vis wind direction. In order to avoid the undesired effects of the turbulent transient flow that occurred at the beginning of the simulation, the results of the first 30 seconds were ignored. To display and compare the wake region at the back of the building, the output images were stored as plan. Table 3 shows the images regarding model simulation and wake region. The lowest wind velocity is close to zero and also, the minimum and maximum velocity are about 6 to 12 m/s, respectively.

Table 3 Numerical simulation of models and comparison of their wake region

	Square		Triangular
S1		T1	
S2		T2	
S3		T3	
S4		T4	
S5		T5	
S6		T6	
S7		T7	
S8		T8	

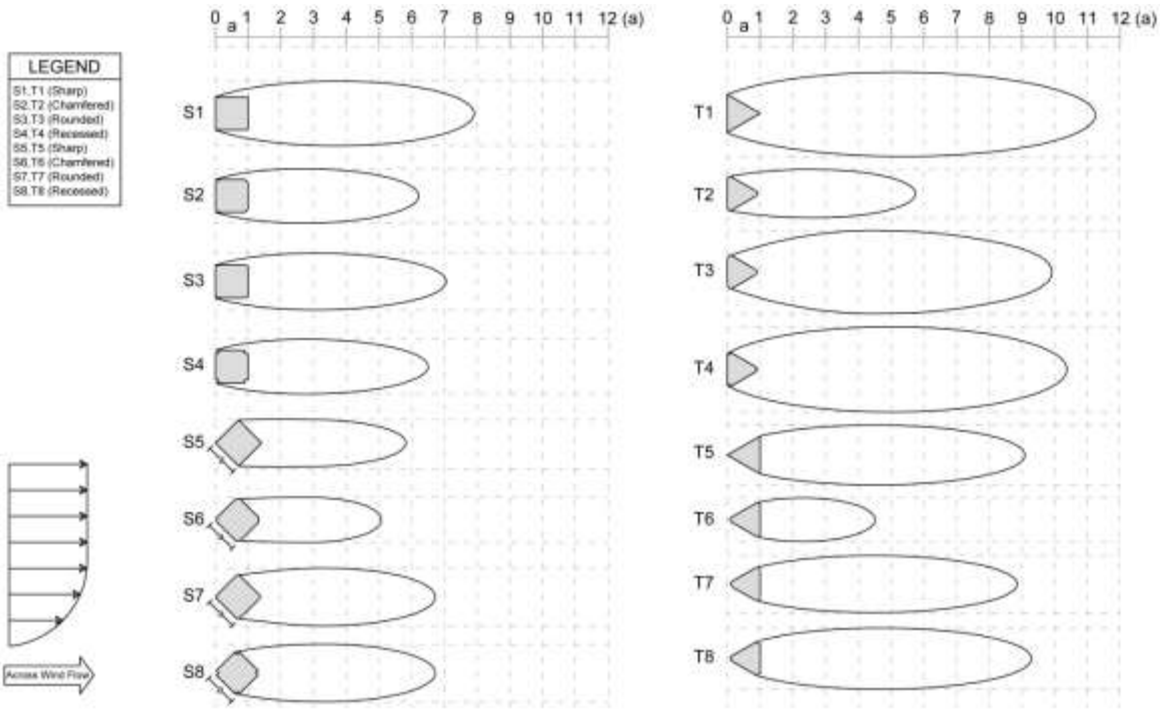


Fig. 4 Model evaluation based on reference models and gridlines

In Table 3 it is observed that the building models have different behavior regarding one another. In order to measure the wake region scientifically, base number (a) and gridlines were defined for each model and the wake region was measured regarding the cross section of each model. The acceptable area could be considered based on table 2 in darker sections where wind speed approaches zero. In Fig. 4, models have been evaluated using base number (a) and gridlines. It should be noted that the base number (a) and the gridlines are defined by the width of the models. Focusing on building aerodynamic modifications, all simulations are carried out using the same wind direction as shown in Fig. 4 and the same wind velocity as indicated in Table 1.

As shown in Fig. 4, in general, models with a triangular footprint generated a longer wake region than the models with square footprint. Under the cross-winds, the models T1 and S1 with wake region lengths of about 11.5a and 8a had the longest wake regions among the models. For the models with Chamfered corners however, the wake regions generated by T2 (Triangular) and S2 (Square) had a length of about 5.5a and 6a respectively. The wake regions generated by triangular and square models with Rounded corners (T3 and S3) had a length of about 10a and 7a respectively, and for the models with corner Recessed, these values were 10.5a (for T4) and 6.5a (for S4) respectively.

The models were then rotated until the wind was aligned with their corner bisector. The changes in the length of wake regions after this rotation can be seen in Fig. 3. For the Sharp-cornered models T5 and S5, the length of wake regions was reduced to 9a and 6a respectively. The length of wake regions generated by the models with Chamfered corners T6 and S6 also saw a change to 4.5a and 5a respectively. For the models T7 and S7 (with Rounded corners), the lengths of wake regions were 9a and 6.5a respectively, and for the models T8 and S8 (with corner Recessed), these lengths were found to be 9.5a and 6.5a respectively. The wake region lengths of all models are shown and compared in Fig. 5.

As shown in Fig. 5 the longest and shortest wake regions were both observed in the models with triangular footprint, more specifically in T1 (Sharp) and T6 (Chamfered corner) respectively.

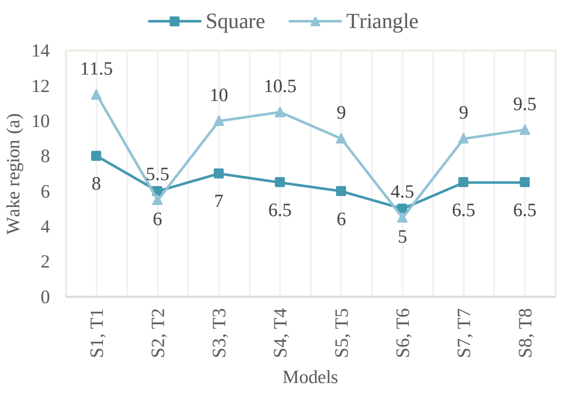


Fig. 5 Comparison of models and their wake region

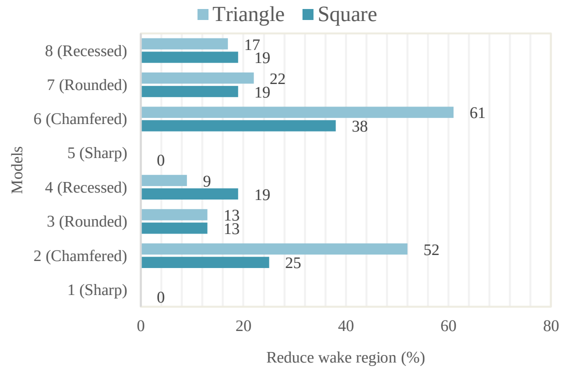


Fig. 6 Wake region reduction in the models with square and triangular footprints comparing to reference Sharp models T1 and S1

In the Following, among the simulated aerodynamic modifications, the most desirable wake region reduction was seen after corner Chamfering, Rounding, and Recessing. Therefore, the sum of the base numbers (a) obtained from simulation results (Fig. 4) is presented to determine the ability of each technique in order to reduce the wake region. In the following, the wake region reductions due to different modifications are compared with the Sharp models (Fig. 6).

Fig. 6 shows the percentage reduction of the wake region of each model relative to each other. On this basis, if Sharp models (T1 and S1) have the lowest rate of reduction which means 0% to reduce the length of the wake region; T1 and S1 number are considered as a whole number so that other model numbers can be compared to them through percentage. So, in the models with the triangular footprint, the wake region reductions achieved by modification are about 52% in T2, 13% in T3, 9% in T4, 22% in T5, 61% in T6, 22% in T7, and 17% in T8. In the following, for the models with the square footprint, the wake region reductions achieved by modification are about 25% in S2, 13% in S3, 19% in S4, 13% in S5, 38% in S6, 19% in S7, and 19% in S8 (all number of models are compared to the Sharp model T1 and S1).

6. Conclusions

Considering the results of simulation, it seems that from a general viewpoint, models having square footprint (S1-S8) could have a better aerodynamic performance in regard to models having triangular footprint (T1-T8) after aerodynamic modifications. In a general classification it seems that shapes of square footprints with Sharp corners (S1) create a smaller wake region back area regarding shapes having triangular footprint with Sharp corners. (T1). Another shape comparison revealed the fact that if Chamfered corner aerodynamic adjusted design is utilized, a noticeable length of the wake region is reduced. The results of simulation showed that Chamfered aerodynamic modification with triangle footprint (T2) has a better aerodynamic performance than the square shape (S1). From the other side, Rounded corner aerodynamic modification holds a weaker performance in regard to Chamfered corner. Considering the results of simulation, aerodynamic Rounded corner modification in model with triangular footprint (T3) shows a weaker performance in regards to the model with square footprint (S3).

In the meantime, Aerodynamic Recessed in models with square footprint (S4) has a better performance in comparison with model with triangular footprint (T4). But in models whose corners are against wind flow (S5-S8, T5-T8), it became evident they had better aerodynamic performance in regard to the previous models (S1-S4, T1-T4). The meaning of aerodynamic performance is the same as the reduction of the wake region. Such that model with square footprint (S5) has a smaller wake region comparing with models with triangular footprint (T5). An important point revealed during simulation was that aerodynamic modification of Chamfered corner whether in models (S2, T2) or (S6, T6) could show favorable performance. This is when the model with triangular footprint (T6) in comparison with mode with square foot print (S6) could develop a smaller wake region. Eventually it seems that aerodynamic modifications of Rounded corner and Recessed in models with square footprints (S7, S3) and model with triangular footprints (T7, T3) have somewhat similar behavior.

Findings of this study showed that between the 16 building models, the model with triangular footprint and aerodynamic modification of Chamfered corner (T6) could perform the best aerodynamically among all other models. It became evident that the model with Sharp corners (T5) have created a wake region 9 times its width, in a general proportion, it could be expressed that the model with triangular footprint and aerodynamic modification of Chamfered corner whose corner is against the wind flow, could reduce the wake region length by 50%. The findings of the survey revealed that the model with square footprint and Chamfered corner aerodynamic modification (S6) exhibited the best performance among all other square models.

It became clear that the model with Sharp corner (S5) formed a wake region 6 times its width. But the model with square footprint and Chamfered modification (S6) formed a wake region 5 times its width. In a general proportion it could be stated that the model with square footprint and Chamfered corner modification whose corner was against wind flow could reduce the wake region up to 30%. It is recommended the design of tall buildings of 50 story (150 meters) and square and triangular footprints with Chamfered corner aerodynamic modification whose corners are against wind flow could have a favorable aerodynamic behavior to decrease the length of wake region.

References

Adamek, K. Vasan, N. Elshaer, A. English, E. and Bitsuamlak, G. (2017), "Pedestrian level wind assessment through city development: A study of the financial district in Toronto", *Sustainable Cities and Society*, **35**, 178-190. https://doi.org/10.1016/j.scs.2017.06.004.

Ali, M. and Armstrong, P. (1995), *Architecture of Tall Buildings*, Council on Tall Buildings and Urban Habitat, New York, McGraw-Hill Book Company.

Amin, J.A. and Ahuja, A.K. (2014), "Characteristics of wind forces and responses of rectangular tall buildings", *Int. J. Adv. Struct. Eng.*., **6**(3), 66.

Amin, J.A. and Ahujab, A.K. (2010), "Aerodynamic modification to the shape of the buildings: a review of the state-of-the-art", *Asian J. Civil Eng. (building and housing)*, **11**(4), 433-450.

Baghaei Daemei, A. (2019), "Wind tunnel simulation on the pedestrian level and investigation of flow characteristics Around buildings", *J. Energy Management Technol.*, **3**(1), 58-68. doi: 10.22109/jemt.2019.135527.1100.

Baghaei Daemei, A., 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 Eng. J.*, https://doi.org/10.1016/j.asej.2018.08.008.

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", *J. Wind Eng. Ind. Aerod.*, **95**(9-11), 941-962. https://doi.org/10.1016/j.jweia.2007.01.013.

Cermak, J.E. (1975), "Applications of fluid mechanics to wind engineering—a Freeman Scholar lecture", *J. Fluids Eng*., **97**(1), 9-38.

Chan, C.M., Chui, J.K.L. and Huang, M.F. (2009), "Integrated aerodynamic load determination and stiffness design optimization of tall buildings", *Struct. Des. Tall Spec. Build.*, **18**(1), 59-80. https://doi.org/10.1002/tal.397.

Dutton, R. and Isyumov, N, (1990), "Reduction of tall building motion by aerodynamic treatments", *J. Wind Eng. Ind. Aerod*., **36**, 739-747. https://doi.org/10.1016/0167-6105(90)90071-J.

Elshaer, A., Gairola, A., Adamek, K. and Bitsuamlak, G. (2017), "Variations in wind load on tall buildings due to urban development", *Sustainable Cities and Society*, **34**, 264-277. https://doi.org/10.1016/j.scs.2017.06.008.

Fadl, M.S. and Karadelis, J.N. (2013), "CFD simulation for wind comfort and safety in urban area: a case study of Coventry university central campus", *Int. J. Architect. Eng. Constr. (IJAEC)*, **2**(2), 131-143.

Flow Design Preliminary Validation Brief (2014), www.simhub.autodesk.com/resources/flow-design-wind-tunnel-validation-brief.

Gu, M. and Quan, Y. (2004), "Across-wind loads of typical tall buildings", *J. Wind Eng. Ind. Aerod.*, **92**(13), 1147-1165. https://doi.org/10.1016/j.jweia.2004.06.004.

Gu, M., Huang, P., Tao, L., Zhou, X. and Fan, Z. (2010), "Experimental study on wind loading on a complicated group-tower", *J. Fluids Struct*., **26**, 1142-1154. https://doi.org/10.1016/j.jfluidstructs.2010.08.003.

Gu, M., Cao, H.L. and Quan, Y. (2014), "Experimental study of across-wind aerodynamic damping of super high-rise buildings with aerodynamically modified square cross-sections", *Struct. Des. Tall Spec. Build.*, **23**, 1225-1245. https://doi.org/10.1002/tal.1137

Günel, M.H. and Ilgin. H.E. (2014), *Tall Buildings Structural Systems and Aerodynamic Form*, First published, Routledge, Taylor & Francis Group, New York.

Hayashida, H. and Iwasa, Y. (1990), "Aerodynamic shape effects of tall building for vortex induced vibration", *J. Wind Eng. Ind. Aerod.*, **33**(1-2), 237-242. https://doi.org/10.1016/0167-6105(90)90039-F.

Holmes, J.D. (2001), *Wind Loading of Structures*, Spon Press, London.

Hu, J.C. Zhou, Y. and Dalton, Y.C. (2006), "Effects of the corner radius on the near wake of a square prism", *Exp. Fluids*, **40**, 106-118.

Ilgin, H.E. and Günel, M.H. (2007), "The role of aerodynamic modifications in the form of tall buildings against wind excitation", *METU JFA*, **24**(2), 17-25.

Irwin, P.A. (1995), "Wind engineering challenges of the new generation of super-tall buildings", *J. Wind Eng. Ind. Aerod.*, **97**(7-8), 328-334. https://doi.org/10.1016/j.jweia.2009.05.001.

Irwin, P.A. and Baker, W.F. (2006), "The Burj Dubai Tower: Wind Engineering", *Structure Magazine*, 28-31.

Irwin, P.A., Kilpatrick, J., Robinson, J. and Frisque, A. (2008), "Wind and tall buildings: Negatives and positives", *Struct. Des. Tall Spec. Build*., **17**, 915-928. https://doi.org/10.1002/tal.482.

Kareem, A., Kijewski, T. and Tamura, Y. (1999), "Mitigation of motion of tall buildings with specific examples of recent applications", *Wind Struct*., **2**(3), 201-251. http://dx.doi.org/10.12989/was.1999.2.3.201.

Karim, A. and Tamura, Y. (1996), "Mitigation of wind-induced motions of tall buildings", *Tall Building Structures: A World View*, Council of Tall Buildings and Urban Habitat, Lehigh University.

Kawai, H. (1998), "Effect of corner modifications on aeroelastic instabilities of tall buildings", *J. Wind Eng. Ind. Aerod*., **74-76**, 719-729. https://doi.org/10.1016/S0167-6105(98)00065-8

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", *J. Wind Eng. Ind. Aerod.*, **96**(10-11), 2093-2103. https://doi.org/10.1016/j.jweia.2008.02.057.

Kim, Y. and Kanda, J. (2010), "Characteristics of aerodynamic forces and pressures on square plan buildings with height variations", *J. Wind Eng. Ind. Aerod*., **98**(8-9), 449-465. https://doi.org/10.1016/j.jweia.2010.02.004.

Kim, Y.C. and Kanda, J. (2013). "Wind pressures on tapered and set-back tall buildings", *J. Fluids Struct*., **39**, 306-321. https://doi.org/10.1016/j.jfluidstructs.2013.02.008.

Kim, Y.C., Kanda, J. and Tamura, Y. (2011), "Wind-induced coupled motion of tall buildings with varying square plan with height", *J. Wind Eng. Ind. Aerod*., **99**(5), 638-650. https://doi.org/10.1016/j.jweia.2011.03.004.

Kwok, K.C.S. (1988), "Effect of building shape on wind-induced response of tall buildings", *J. Wind Eng. Ind. Aerod*., **28**(1-3), 381-390 .https://doi.org/10.1016/0167-6105(88)90134-1.

Kwok, K.C.S. and Bailey, P.A. (1987), "Aerodynamic devices for tall building and structures", *J. Eng. Mech. – ASCE*, **4**(111), 349-365.

Kwon, D.K. and Kareem, A. (2013), "Comparative study of major international wind codes and standards for wind effects on tall buildings", *Eng. Struct.*, **51**, 23-35. https://doi.org/10.1016/j.engstruct.2013.01.008.

Liu, X., Niu, J. and Kwok, K.C.S. (2013), "Evaluation of RANS turbulence models for simulating wind-induced mean pressures and dispersions around a complex-shaped high-rise building", *Building Simulation*, **6**(2), 151-164.

Melbourne, N.H. and Cheung, J.C.K. (1988), "Designing for serviceable accelerations in tall buildings", *Proceedings of the 4th International Conference on Tall Buildings*, Hong Kong and Hanghai, China.

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", *J. Wind Eng. Ind. Aerod*., **50**, 319-328. https://doi.org/10.1016/0167-6105(93)90087-5.

Mochida, A. and Lun, I.Y.F. (2008), "Prediction of wind environment and thermal comfort at pedestrian level in urban area", *J. Wind Eng. Ind. Aerod*., **96**(10-11), 1498-1527. https://doi.org/10.1016/j.jweia.2008.02.033.

Parker, D. and Wood, A. (2013), *The Tall Buildings Reference Book*, First published, Routledge, Taylor & Francis Group, USA.

Schueller, W. (1977), *High-Rise Building Structures*, John and Wiley Sons Inc., New York.

Scott, D., Hamilton, N. and Ko, E. (2005), "Structural design challenges for tall buildings", *Structure Magazine*. **2**, 20-23.

Sharma, A., Mittal, H. and Gairola, A. (2018), "Mitigation of wind load on tall buildings through aerodynamic modifications: Review", *J. Build. Eng.*, **18**, 180-194. https://doi.org/10.1016/j.jobe.2018.03.005.

Shiqing, H., Xuhui, H. and Hua, H. (2017), "Characteristics of the aerodynamic interference between two high-rise buildings of different height and identical square cross-section", *Wind Struct*., **24**(5), 501-528. https:// doi.org/10.12989/was.2017.24.5.501.

Sullivan, P.P., McWilliams, J.C. and Moeng, C.H. (1994), "A subgrid-scale model for large-eddy simulation of planetary boundary-layer flows", *Bound.-Lay. Meteorol.*, **71**(3), 247-276.

Tamura, T. and Miyagi, T. (1999), "The effect of turbulence on aerodynamic forces on a square cylinder with various corner shapes", *J. Wind Eng. Ind. Aerod*., **83**(1-3), 135-145. https://doi.org/10.1016/S0167-6105(99)00067-7.

Tamura, T., Miyagi, T. and Kitagishi, T. (1998), "Numerical prediction of unsteady pressures on a square cylinder with various corner shapes", *J. Wind Eng. Ind. Aerod*., **74**(76), 531-542. https://doi.org/10.1016/S0167-6105(98)00048-8.

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", *J. Wind Eng. Ind. Aerod*., **107**(108), 179-191. https://doi.org/10.1016/j.jweia.2012.04.014.

Tanaka, H., Tamura, Y., Ohtake, K., Nakai, M., Chul, K.Y. and Kumar, B.E. (2013), "Aerodynamic and flow characteristics of tall buildings with various unconventional configurations", *Int. J. High-Rise Build.*, **2**(3), 213-228.

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", *J. Wind Eng. Ind. Aerod*., **96**(10-11), 1749-1761. https://doi.org/10.1016/j.jweia.2008.02.058.

Wahrhaftig, D.M.A. and da Silva, M.A. (2018), "Using computational fluid dynamics to improve the drag coefficient estimates for tall buildings under wind loading", *Struct. Des. Tall Spec. Build.*, **27**(3), e1442. https://doi.org/10.1002/tal.1442

Wille, R. (1960), "Kármán Vortex Streets", *Adv. Appl. Mech.*, **6**, 273-287.

Xie, J. (2012), "Aerodynamic optimization in super-tall building designs", *Proceedings of the 7th International Colloquium on Bluff Body Aerodynamics and its Applications (BBAA7).* Shanghai, China; September: 2-6.

Xie, J. (2014), "Aerodynamic optimization of super-tall buildings and its effectiveness assessment", *J. Wind Eng. Ind. Aerod*., **130**, 88-98. https://doi.org/10.1016/j.jweia.2014.04.004.

Xu, Z. and Xie, J. (2015), "Assessment of across-wind responses for aerodynamic optimization of tall buildings", *Wind Struct*., **21**(5), 505-521. http://dx.doi.org/10.12989/was.2015.21.5.505.

Yasa, E. (2016), "Computational evaluation of building physics—The effect of building form and settled area", microclimate on pedestrian level comfort around buildings", *Building Simulation*. **9**(4), 489-499.

Yechuri, N.V., Kumar, M.S., Raju, V.D. and Kumar, P.R. (2015), "Aerodynamic Studies of non – Circular High – rise buildings", *J. Mech. Civil Eng. (IOSR-JMCE)*, **12**(5), 14-21.

Zhi, L., Chen, B. and Fang, M. (2015), "Wind load estimation of super-tall buildings based on response data", *Struct. Eng. Mech*., **56**(4), 625-648. http://dx.doi.org/10.12989/sem.2015.56.4.625.

Zhiyin, Y. (2015), "Large-eddy simulation: Past, present and the future", *Chinese J. Aeronautics*, **28**(1), 11-24.

https://doi.org/10.1016/j.cja.2014.12.007.

CC