PAPER • OPEN ACCESS

Investigation of the Effect of the Wind Speed on the Aerodynamic and Architectural Design of Tall Buildings

To cite this article: Yong Win Hui *et al* 2023 *J. Phys.: Conf. Ser.* **2523** 012039

Investigation of the Effect of the Wind Speed on the Aerodynamic and Architectural Design of Tall Buildings

Yong Win Hui[1, a)], Abdulkareem Sh. Mahdi Al-Obaidi[1, b)], Tamil Salvi Mari[2, c)], Sujatavani Gunasagaran[2, d)], Michael Ching[3, e)]

[1]School of Engineering, Taylor's University, Malaysia
[2]School of Architecture, Building and Design, Taylor's University, Malaysia
[3]CH&I Architecture Sdn. Bhd., Malaysia

[a)]Corresponding Author: winhui.yong@sd.taylors.edu.my
[b)]abdulkareem.mahdi@taylors.edu.my
[c)]tamilsalvi.mari@taylors.edu.my
[d)]sujatavani.gunasagaran@taylors.edu.my
[e)]ch@chiarch.com.my

Abstract. The magnitude, speed, direction, and distribution of the wind are known to have a negative impact on high-rise buildings, particularly when the building is extremely tall. As the height of the building increases, the effect of the wind increases significantly. This results in a high wind pressure exerted on the building which causes the wind loads subjected by the buildings to increase considerably. Thus, this study investigates the wind effect on tall buildings and proposes a solution to reduce the aerodynamic drag through a design that meets both architectural and aerodynamic considerations. Investigation of wind effect on tall buildings with different wind speeds, heights and shapes is carried out. Numerical methods are utilized to study the wind pressure, drag coefficient and pressure distribution on different types of buildings. ANSYS Fluent is used for computational fluid dynamics (CFD) simulation to assess the drag coefficient of the buildings. The drag coefficient of the building is determined in Reynold's number range of 1×10^6 to 2×10^6. The study is conducted in low subsonic speed (Ma < 0.3). The result shows that the drag coefficient of the building tends to be constant at low subsonic speeds. Besides, the study found that the drag coefficient increases by 1% to 3% for every 0.1 m increment in the height of the building. The study also found that a cylindrical building has the lowest drag coefficient because of its more streamlined shape. Overall, this study provided guidance and recommendations for wind resistance which can be taken into consideration when designing tall buildings. Hence, the building can be built even taller while maintaining its rigidity and stability.

Keywords: CFD simulation, Drag, Tall buildings, Wind.

1. Introduction

The development of tall buildings is increasing worldwide, accommodating rapid urbanization around the world. The main structure and façade of these tall buildings must be built to withstand the severe winds to which they will be subjected throughout their estimated life. It is crucial to determine the aerodynamic loads, especially the drag, due to the wind speed, wind magnitude, direction, and wind load distribution on tall buildings to identify the design factor for shaping and positioning the buildings.

As the height of the building increases, the wind effect on these supertall buildings increases as well. As strong winds blew around these buildings, vortices appeared on the other side of the buildings as illustrated in figure 1 and it forms in a process known as vortex shedding. Vortex shedding is caused by the flow separation on the edges of the body. This will result in low pressure regions which create a suction force that causes vibration in the building. This process acts differently depending on the drag coefficient and how streamlined the building design is. The lower the drag coefficient of the building, the lower the wind pressure exerted on the building. Therefore, if the building has a lower drag coefficient, there will be less wind effect acting on the buildings. Thus, the drag coefficient of a building should be as low as possible to ensure its rigidity of the building.

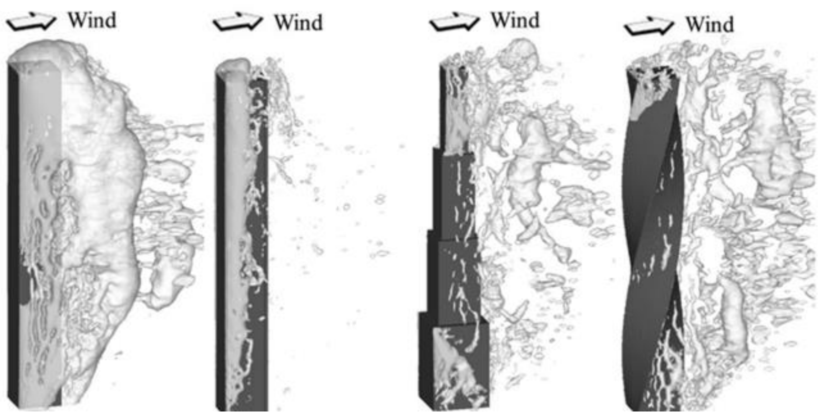


Figure 1. Vortex shedding on buildings [1].

Numerous studies on the aerodynamics of tall buildings have been conducted by researchers to improve the development of supertall buildings in the future. One of the most noteworthy studies has been carried out by Tanaka et al. [1, 2] on the investigation of the effect of aerodynamic forces and wind pressure on tall buildings with unconventional configurations using wind tunnel and computational fluid dynamics (CFD). Large-Eddy Simulation (LES) model was used to solve the numerical calculations in this study. The building configurations include corner modification, tilted, tapered, helical, openings and composite. The research found that the 180° helical square model is the most ideal modification of the buildings because it creates the weakest vortex.

Kareem et al. [3] carried out a study to evaluate the performance of Canton Tower during typhoon events by determining the wind characteristics and structural responses using a wind tunnel. The actual performance of the full-scale Canton Tower under the winds was investigated and compared with the wind tunnel predictions. The study has found that the tower serviceability during typhoon events is satisfied if a 90% confidence interval is used for prediction.

An assessment done by Xie [4] summarized the analysis of optimization on super-tall buildings and the aerodynamic effectiveness of tapering, twisting and stepping in super-tall building design using a wind tunnel. It was found that across-wind responses can be reduced by using tapering and stepping on the design of buildings. The study revealed that although the aerodynamic efficiency of twisting generally improves as the twisting level increases, the increment of effectiveness tends to decrease and there is a maximum reduction limit.

Elshaer et al. [5] presented an aerodynamics optimization procedure on the corner of rectangular building to reduce wind load. Artificial Neural Network (ANN) based surrogate model was used in the study to evaluate the optimization procedure. The study has found that the drag coefficient of the optimal shape is 30% lower than the sharp edge corner. A similar study was carried by Neethi and Elsa [6] to determine wind excitation subjected on rectangular building's corner geometry, sculptured building tops and vertical openings with height of 150 m by using ANSYS Fluent for CFD simulation. They concluded that corner modifications such as chamfered corners, rounded corners and tapered corners can reduce wind induced response of buildings significantly.

Daemei et al. [7] determined the drag coefficient performance of aerodynamics modification on a triangular building with a building height of 120 m by using Autodesk Flow Design 2014 for CFD simulation. It is concluded that rounded corner aerodynamic modification and tapered aerodynamic formation should be applied to buildings to reduce the drag coefficient. Moreover, Kode et al. [8] carried out a study to identify the pressure distribution and pressure coefficient of the Tech Park using ANSYS Fluent. They found that the pressure distribution in the cube and cuboid is uniform but the pressure distribution on uneven facades is nonuniform. Besides, the flow is found to be more turbulent with the presence of the roof and windows on the side facades.

Germi and Kalehsar [9] evaluated the upstream and downstream interference effect on principal and isolated rectangular buildings using LES turbulence model in CFD simulation. The result shows that the mean drag coefficient of the isolated building is higher than principal buildings in most of the interference cases. The critical velocity of the wind will increase for upstream interference cases which reduces the possibility of the occurrence of aeroelastic instabilities.

In Malaysia, Shafii and Othman [10] discussed the climatic and wind speed distribution on low-rise and high-rise buildings around Malaysia. The estimation of wind loading for structural design in Malaysia is based on MS1553: 2002 Code of Practice on Wind Loading for Building Structures [11]. They concluded that buildings in Malaysia are often unaffected by extreme wind events due to the low wind speed and corresponding low wind loading. Other than that, a study was carried out by Nizamani et al. [12] to investigate the effects of wind load on a fifteen-storey high-rise building using the Malaysian Standard Code of Practice on Wind Loading for Building Structure. The study found that increasing the wind speed significantly increases the storey resultant forces, namely storey shear force and moment.

Although there have been numerous studies on aerodynamic wind loads and optimization on buildings, most of the studies are based on rectangular and triangular shape buildings. There is limited study focused on investigating wind effect on cylindrical shape buildings. One study that comes close to a cylindrical shape building is the study done by Kareem et al. [3] to evaluate the performance of Canton Tower, an elliptical shape building. Similarly, Tanaka et al. [1, 2] evaluated the performance of elliptical shape buildings against wind effects. Besides, most of the studies only investigated the aerodynamic performance of building at a constant height. There is still lack of investigation on aerodynamics drag on building with different heights.

Thus, the purpose of the present study is to investigate the effect of wind speed, its direction and distribution on the aerodynamic and architectural design of tall buildings. The building selected for the investigation is the Citibank Tower which is located in Kuala Lumpur, Malaysia. The aerodynamics performance of the Citibank Tower was investigated by numerical method which uses ANSYS Fluent to estimate the drag coefficient of the building at different wind speeds. The model of Citibank Tower is also generated into different heights to investigate the effect of height on the drag coefficient. Lastly, the wind effect on buildings with different shapes was investigated as well.

2. Research methodology

2.1. Drag force of buildings

The drag force subjected by the building is calculated using equation (1):

$$F_D = C_D \frac{1}{2} \rho V^2 S_{ref} \tag{1}$$

where F_D is the drag force, C_D is the drag coefficient, ρ is the density of the fluid, V is the wind velocity and S_{ref} is the reference area.

At subsonic speeds, the total drag is consisting of skin friction drag and pressure drag as given in equation (2):

$$C_D = C_{D_{fr}} + C_{D_p} \tag{2}$$

where C_D is the drag coefficient, $C_{D_{fr}}$ is the skin friction drag and C_{D_p} is the pressure drag. The induced drag is not considered because the building is a bluff body which generate no lift.

For Reynold's number between the range of 5×10^5 to 5×10^7, the skin friction drag is calculated using equation (3) [13]:

$$C_{D_{fr}} = \frac{0.074}{{Re_L}^{\frac{1}{5}}} \tag{3}$$

where $C_{D_{fr}}$ is the skin friction drag and Re_L is the Reynold's number based on length of model.

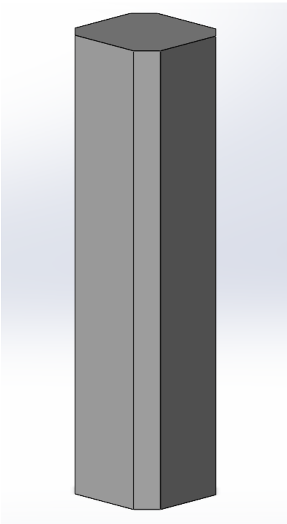
The difference between skin friction drag and pressure drag is that skin friction drag is caused by wall shear stress acting between the air and the building surface due to viscosity and surface roughness of the building. While pressure drag is caused by the wind load acting normal to the surface of the building.

For the drag calculations of buildings, only these two components of drag are considered. The assumption used here is that when the Mach number is below 0.3, the flow is incompressible.

2.2. *Building geometry*

2.2.1. Investigation on the effect of wind speed. As mentioned earlier, the building in Kuala Lumpur was chosen for the investigation on the effect of wind speed is the Citibank Tower with a height of 190.2 m and a typical floor area of 1608.06 m^2 [14] as shown in figure 2(a). Citibank Tower was chosen due to its simple geometry, and it is located at the urban areas. Most importantly, it is a real existing tower which its dimensions are managed to be found. It is considered as a sample of building and other building can be considered using similar approach used in this paper. The scale model of the Citibank Tower was created as shown in figure 2(b) by using SOLIDWORKS 2021. Table 1 presents the dimensions of the Citibank Tower model.

(a)



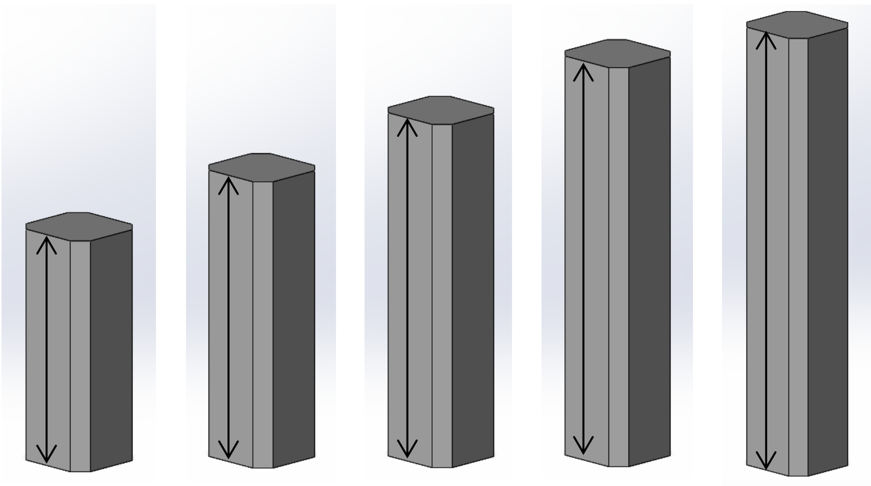
(b)

Figure 2. Citibank Tower (a) Actual. [14] (b) Modelled.

Table 1. Dimensions of Citibank Tower test model.

Length (m)	Width (m)	Height (m)	Chamfer length (m)	Chamfer angle (°)	Floor area (m^2)
0.1463	0.1519	0.6	0.0338	135	0.021073

2.2.2. Investigation on the effect of height. The height of the test model of the Citibank Tower is manipulated into 0.4 m, 0.5 m, 0.7 m and 0.8 m as shown in figure 3 to investigate the effect of wind on buildings with different heights.



0.4 m 0.5 m 0.6 m 0.7 m 0.8 m

Figure 3. Citibank Tower with different heights.

2.2.3. Investigation on the effect of shape. For shape investigation, the building model is manipulated into square, cylindrical and triangular shape while keeping the same floor area as the scaled Citibank Tower which is 0.021073 m^2 and constant building height which is 0.6 m as shown in figure 4.

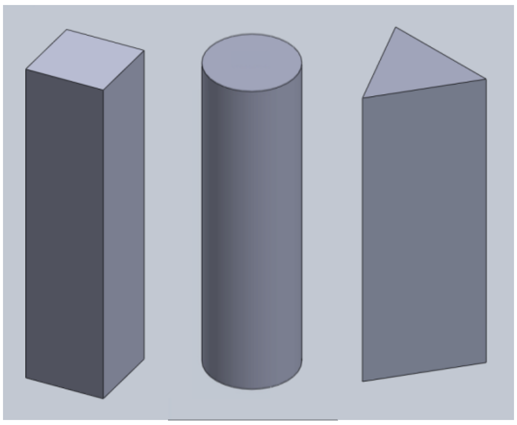


Figure 4. Square, cylindrical and triangular buildings.

2.3. Numerical method

In order to investigate the effect of wind speed on the tall buildings, computational fluid dynamics (CFD) is utilized to simulate the wind flow around the tall buildings. Drag coefficient, wind pressure and wind load distribution on the buildings are solved by Finite Volume Method in ANSYS Fluent.

2.3.1. Enclosure domain. In order to simulate the building in wind tunnel, the enclosure domain was set based on the recommendation by Franke et. al. [15]. Franke et. al. recommended that the blockage ratio in the enclosure domain should not exceed 3%. Thus, the distance between the building model and the inlet is set to 5H while the outlet is set to 15H as illustrated in figure 5(a). Besides, the distance between the top of the domain and the top of the building is set to 5H as shown in figure 5(b). The same enclosure domain was utilized in the study on aerodynamics of typical high-rise building carried out by Kode et. al. [8] which has proven the reliability of this setting of domain. Another domain is created close to the building model to capture the flow around the building. The length and width of the domain are set to H while the height is set to 1.5H as shown in figure 5(b). 'H' represented the height of the building model. The air is flowed from the +Z-direction to the -Z-direction.

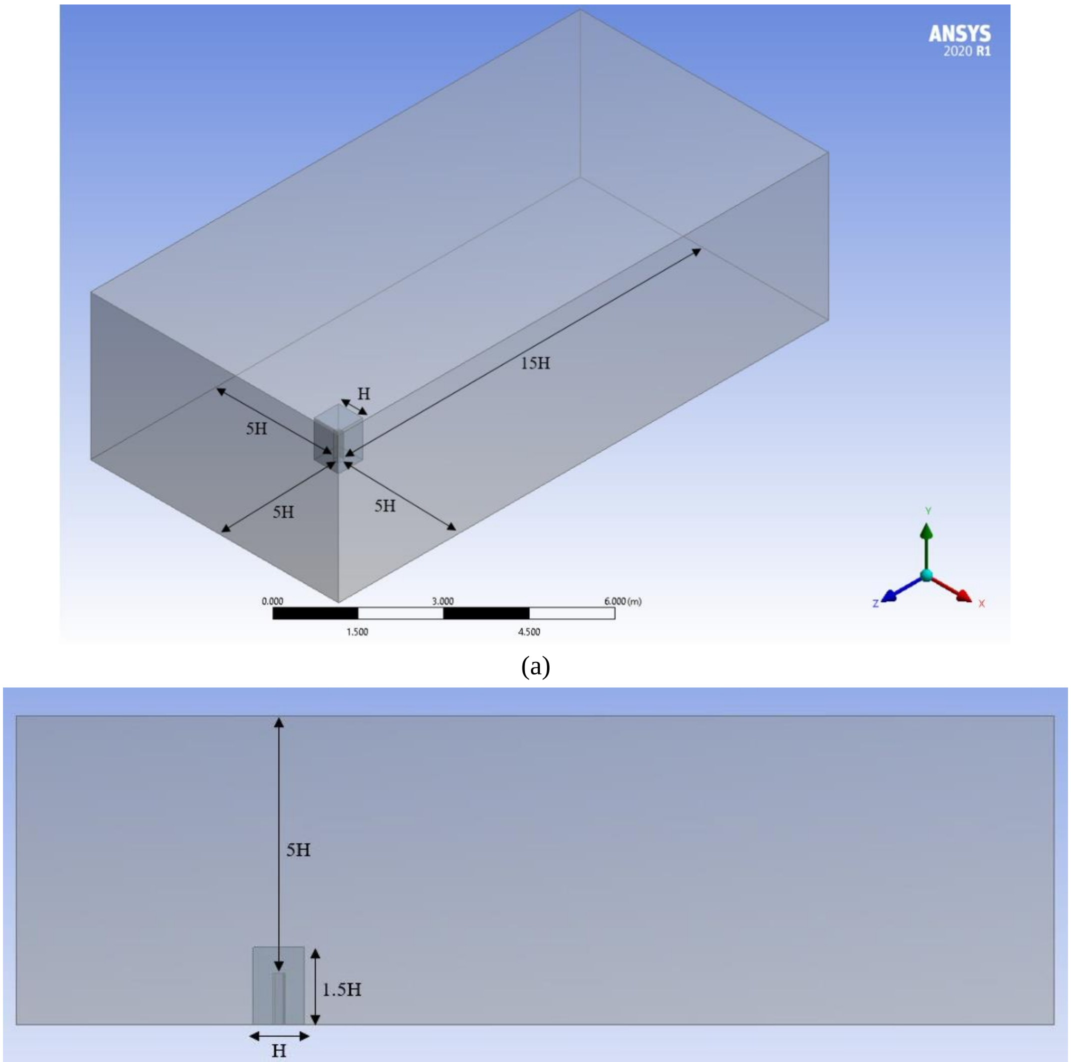


Figure 5. Enclosure domain based on Franke's Recommendations (a) Isometric view. (b) Side view.

2.3.2. Meshing. Meshing is a significant component to capture the boundary layer and ensure the accuracy of simulation results. The mesh quality can be evaluated by checking the skewness of the meshing. The mesh quality is considered high when the skewness is nearer to zero. In this study, a body of influence is applied to the area close to the building model with the element size of 0.01 m. Finer mesh is necessary to be applied in the area around the building and in the wake region to keep track of the flow separation at the building's sharp corner and also to visualize the vortex shedding better [9].

Apart from that, inflation layers are added around the building to capture the boundary layer better during the simulation. The first layer thickness of the inflation is set to 0.00001 m with a maximum layer of 12 at the growth rate of 1.2. The generated mesh is shown in figures 6 and 7.

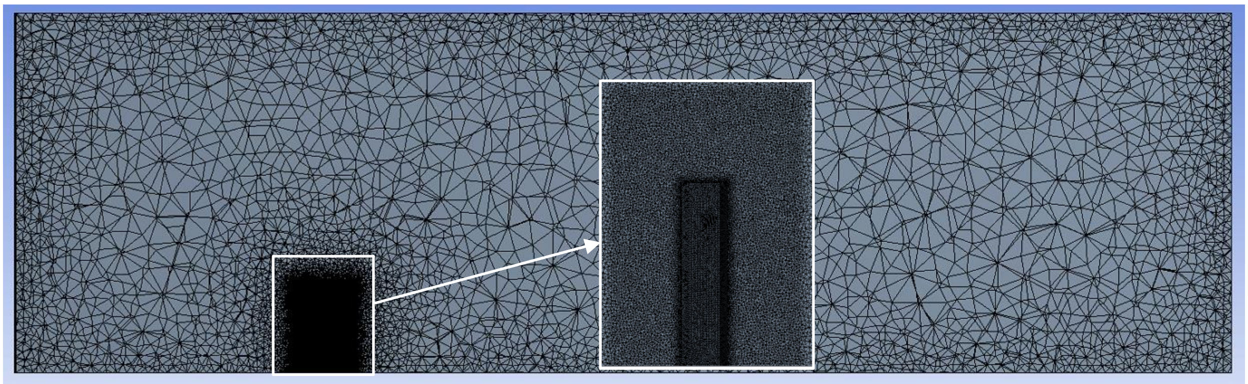


Figure 6. Meshing on the enclosure domain and building.

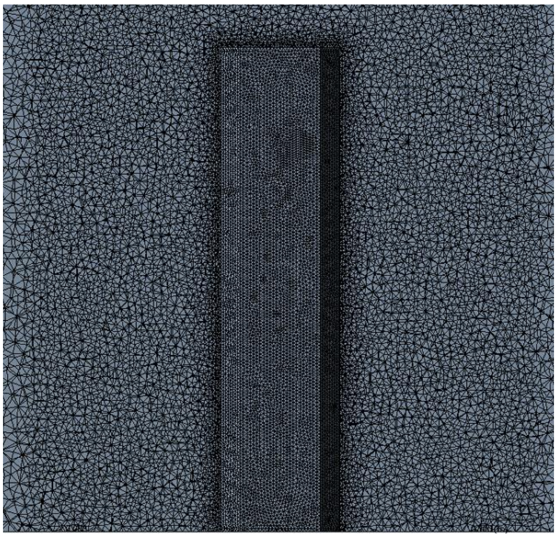


Figure 7. Mesh refinement applied on the building.

Besides that, mesh refinement is implemented by applying face sizing on the surface of the building. Grid independence test is carried out to select the appropriate face sizing for the building surface. Although smaller mesh size gives more accurate results, it requires longer solving time. Thus, grid independence test is important to be used to find a balance between result accuracy and solving time. A grid independence test is carried out on the face sizing of the Citibank Tower model. Based on the graph in figure 8, the face sizing of 0.004 m (997741 nodes) is chosen because the face sizing of 0.002 m (2115089 nodes) started to give insignificant changes in drag coefficient and required longer solving time. The mesh configuration is summarized in Table 2.

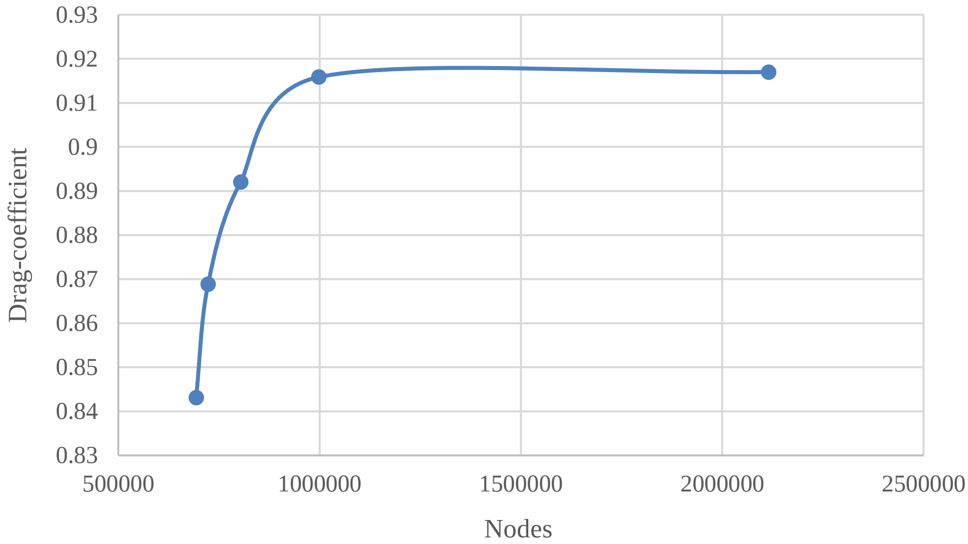


Figure 8. Grid independence test on face sizing.

Table 2. Mesh configurations.

Body of influence (m)	Body element size (m)	Face sizing (m)	Inflation first layer thickness (m)
H (X) × 1.5H (Y) × H (Z)	0.01	0.004	0.00001

2.3.3. Solver. Since Mach number in this study is less than 0.3, the flow is considered incompressible flow, so the pressure-based solver is used for the simulation. An investigation was carried out to study the drag coefficient of cylindrical building using different turbulence models in CFD simulation. The drag coefficient of the cylindrical building is determined in Reynold's number range of 1×10^6 to 2×10^6. Based on figure 9, the bar chart shows that the drag coefficients obtained by the Spallart-Allmaras (SA), SST k-omega (k-ω) and Standard k-ω turbulence models are closest to 0.3 which is the reference drag coefficient of cylinder. However, SA was chosen for the simulation as it is time efficient, and it has the lowest computational cost among all 5 turbulence models. Most importantly, SA is accurate for wall-bounded flows which suit for the present study as the effect near the wall of the building model is significant to be captured to obtain accurate result. The reading of drag coefficient is taken when the solution is converged which can be identified when the graph as shown in figure 10 started to be constant.

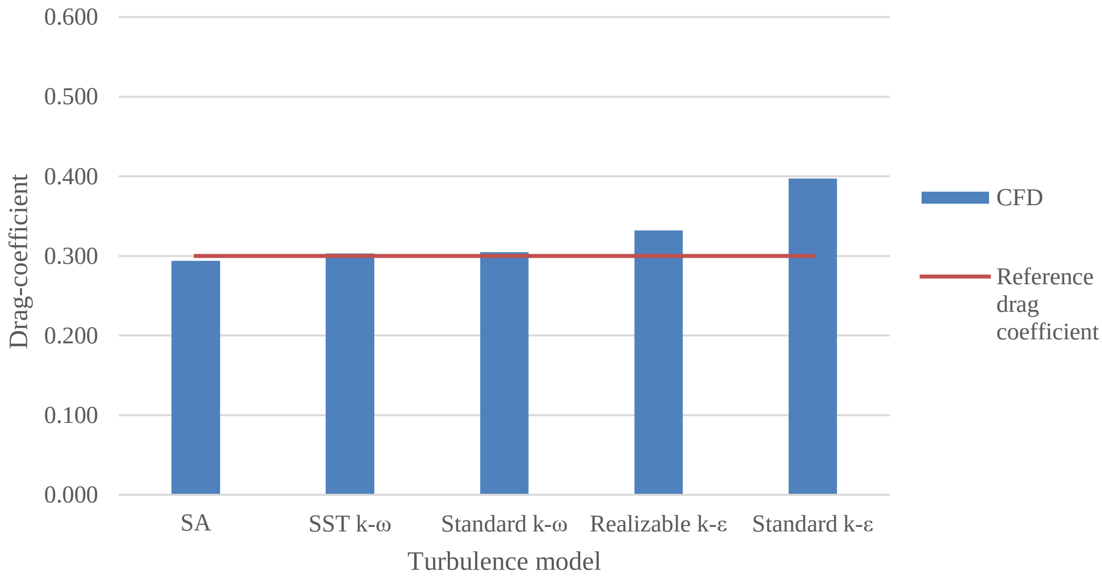


Figure 9. Comparison of drag coefficient of cylindrical building using different turbulence models.

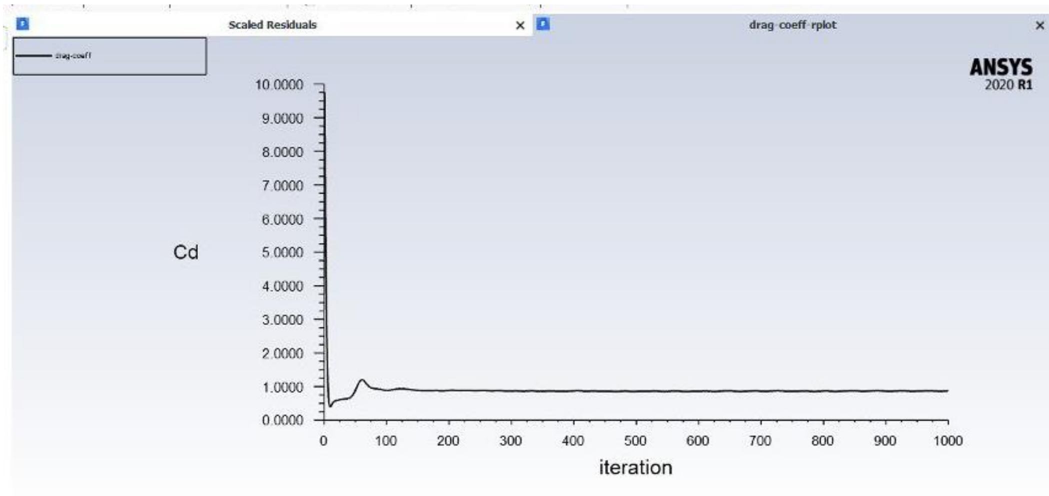


Figure 10. Solution is converged.

3. Results and discussion

3.1. Validation of numerical results

This section discusses on the results obtained from aerodynamics investigation on buildings with different heights, different shapes and at different wind speeds. In order to validate the result of numerical calculation, figure 11 shows the comparison of drag coefficient of two standard shapes which are cube and cylinder with the existing value of drag coefficient in reference. Based on Table 3, it shows that cube obtained the maximum error which is 12.12% while for the cylinder is 1.38%. This is to be expected because the shape of the cube is having more resistance compared to cylinder which is more streamlined. Therefore, the numerical settings in term of meshing and solver are proved to be valid for the air flow simulation. The exact same settings were applied on the simulation to investigate the effect of wind speed, height and shape of building on the drag coefficient of the building.

Table 3. Drag coefficient of models with different shapes.

Shape	Drag-coefficient		Percentage
	CFD	Reference	of error (%)
Cube	1.177	1.05 [13]	12.12
Cylinder	0.296	0.30 [13]	1.38

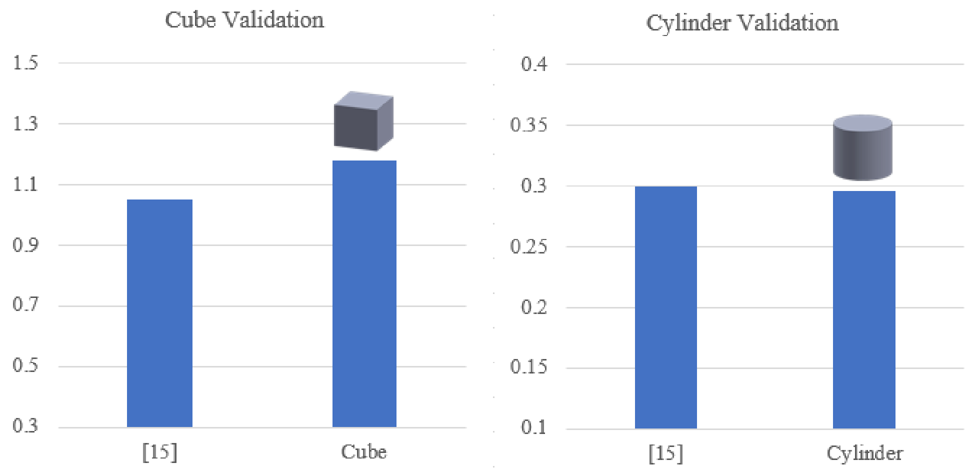


Figure 11. Validation of drag coefficient of different shapes with reference.

3.2. *Investigation on the effect of wind speed*

The first case is to study the effect of wind speed on the test model of Citibank Tower with a model height of 0.6 m. The results obtained from CFD simulation for the investigation on the effect of wind speed are presented in figure 12. Based on the graph, the drag coefficient is almost constant with the speed, which agrees with most of the results published regarding the effect of wind speed on the drag coefficient such as Al-Obaidi and Lai [16]. The speed of the wind is very low subsonic speed where the air can be considered incompressible, meaning there is no change in density.

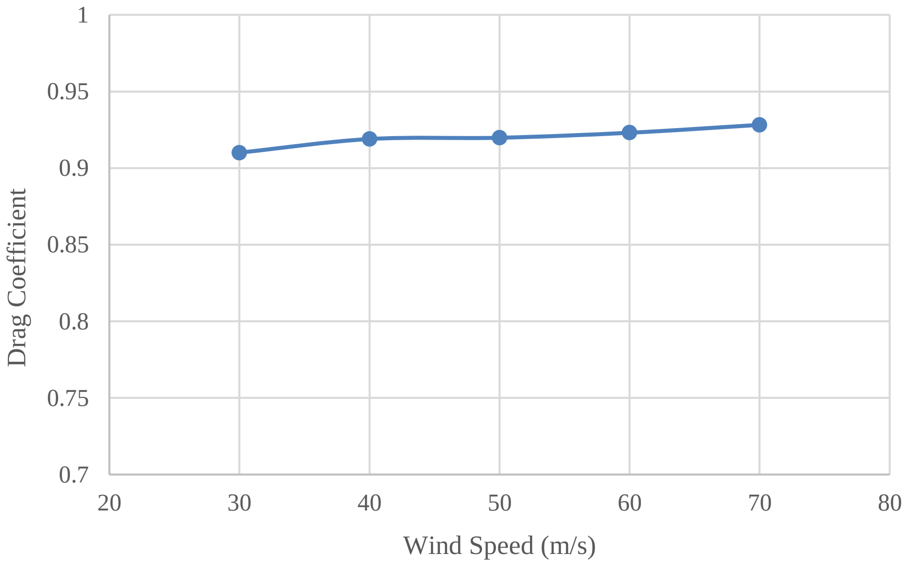


Figure 12. Comparison of drag-coefficient for Citibank Tower at different wind speeds.

3.3. Investigation on the effect of height

The second case is to study the effect of height of the Citibank Tower at constant wind speed of 50 m/s. Figure 13 shows the drag coefficient of Citibank Tower with the range of height from 0.4 to 0.8 m with 0.1 m interval. The result shows that the drag coefficient is affected by the height of the Citibank Tower. For every 0.1 m increment in the height of the building, the drag coefficient increases by 1% to 3%. At constant wind speed, the higher the Citibank Tower, the higher the drag coefficient. This is because of the increasing in frontal area of the building due to increasing of height of building. Increasing in frontal area causes the increase in both skin friction drag and pressure drag and thus the drag coefficient increases.

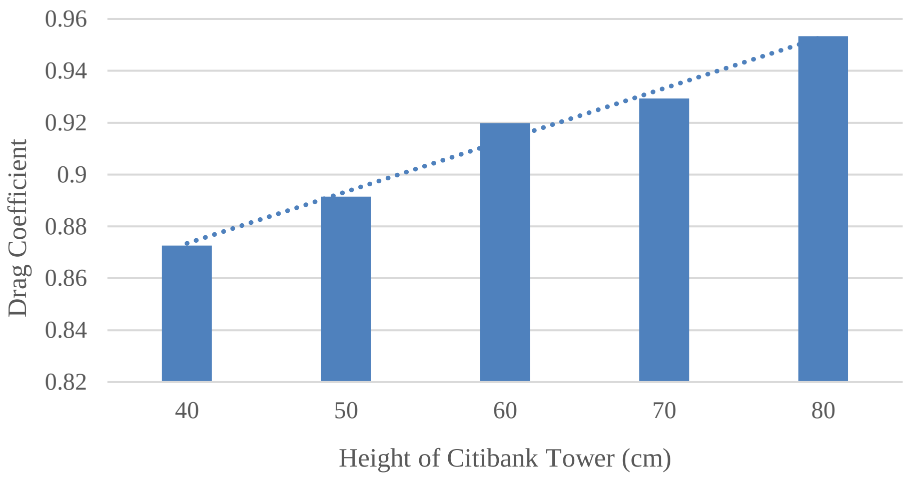


Figure 13. Comparison of drag-coefficient for Citibank Tower with different heights.

3.4. Investigation on the effect of shape

The third case is to study the effect of shape of the building with constant floor area at constant wind speed of 50 m/s. Table 4 shows the results obtained from CFD for square, cylindrical and triangular buildings. Triangular building has the highest percentage of error when comparing to the reference drag coefficient which is 5.7% while square building has the lowest percentage of error which is 1.93%. Figure 14 shows the comparison of drag coefficient of square, cylindrical and triangular building with the reference drag coefficient of its corresponding shape. The drag coefficient of all three shapes of buildings is summarized in figure 15. The figure shows that square building has the highest drag coefficient while cylindrical building has the lowest drag coefficient. This is because the square shape is having more resistance compare with cylindrical shape which is more streamlined.

Table 4. Drag coefficient of tall buildings with different shapes.

Shape	Drag-coefficient		Percentage
	CFD	Reference	of error (%)
Square	1.275	1.30 [17]	1.93
Cylindrical	0.294	0.30 [13]	1.99
Triangular	0.830	0.88 [17]	5.70

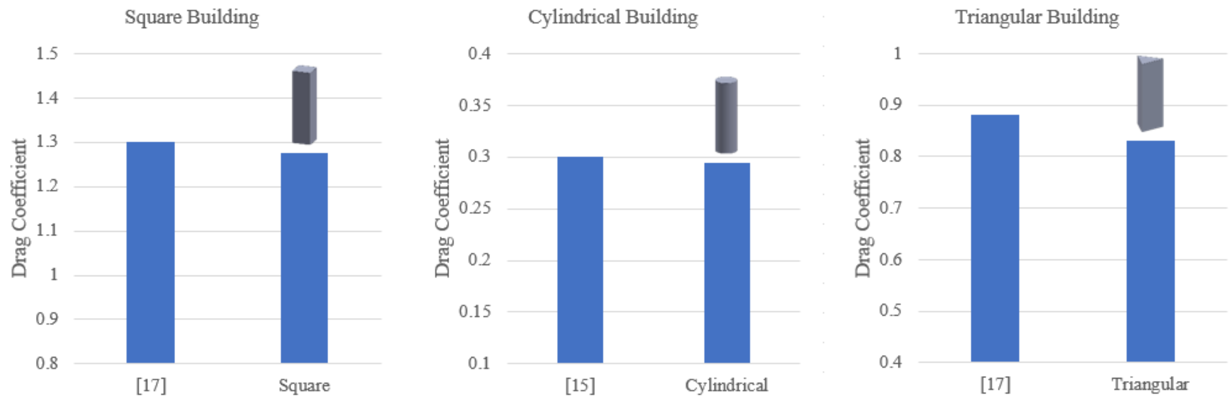


Figure 14. Comparison of drag coefficient of different shapes with references.

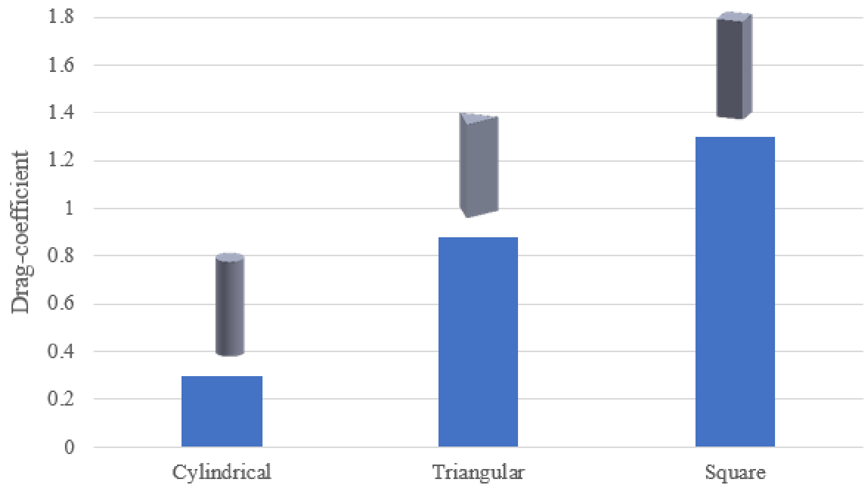


Figure 15. Comparison of drag coefficient of different shapes.

Figure 16 depicts the streamlines over the cylindrical, triangular and square buildings which visualized the conditions of vortex shedding around and behind the buildings. Vortex shedding is occurred due to the flow separation at the edges of the building. In order to quantify the vortex shedding, the pressure distribution along the height of square, triangular and cylindrical buildings on the upstream and downstream are captured and plotted as shown in figure 17. While figure 18 shows the pressure difference between the upstream and downstream of the buildings. Based on figure 18, square building has the largest pressure difference between the upstream and downstream flows compared to triangular and cylindrical buildings. This means that square building has the greatest pressure drops from its upstream to downstream and causes largest area of the vortex shedding as shown in figure 16(c). Vortex shedding results in low pressure regions which create a suction force that causes vibration in the building. The cylindrical building, however, has the lowest pressure difference and thus having the least vortex shedding as shown in figure 16(a). This is because cylindrical building is more streamlined compared to triangular and square building. Thus, cylindrical building has the least vortex shedding and causes less vibration in the building compared to square and triangular buildings. Besides, figure 19 shows the pressure distribution over the investigated shapes of buildings. The pressure is highest on the windward surface for cylindrical, triangular and square building. This is due to deceleration of wind speed on the at the windward surface of the buildings.

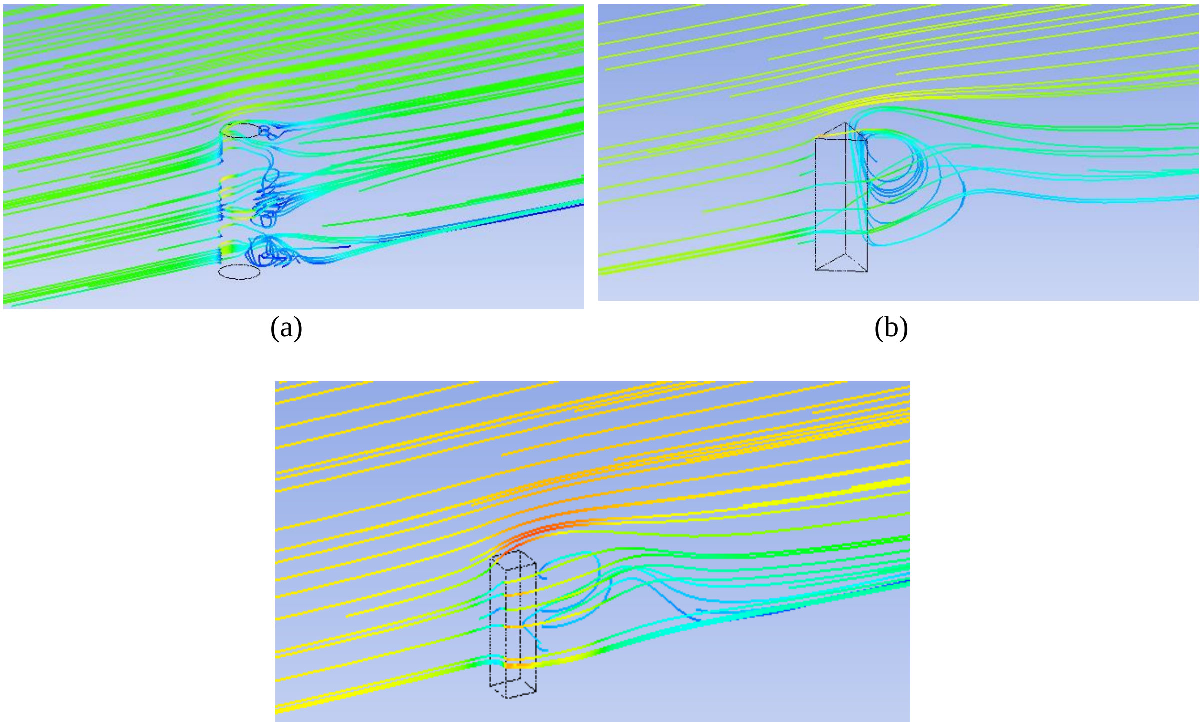


Figure 16. Streamline on different shapes of buildings (a) Cylindrical. (b) Triangular. (c) Square.

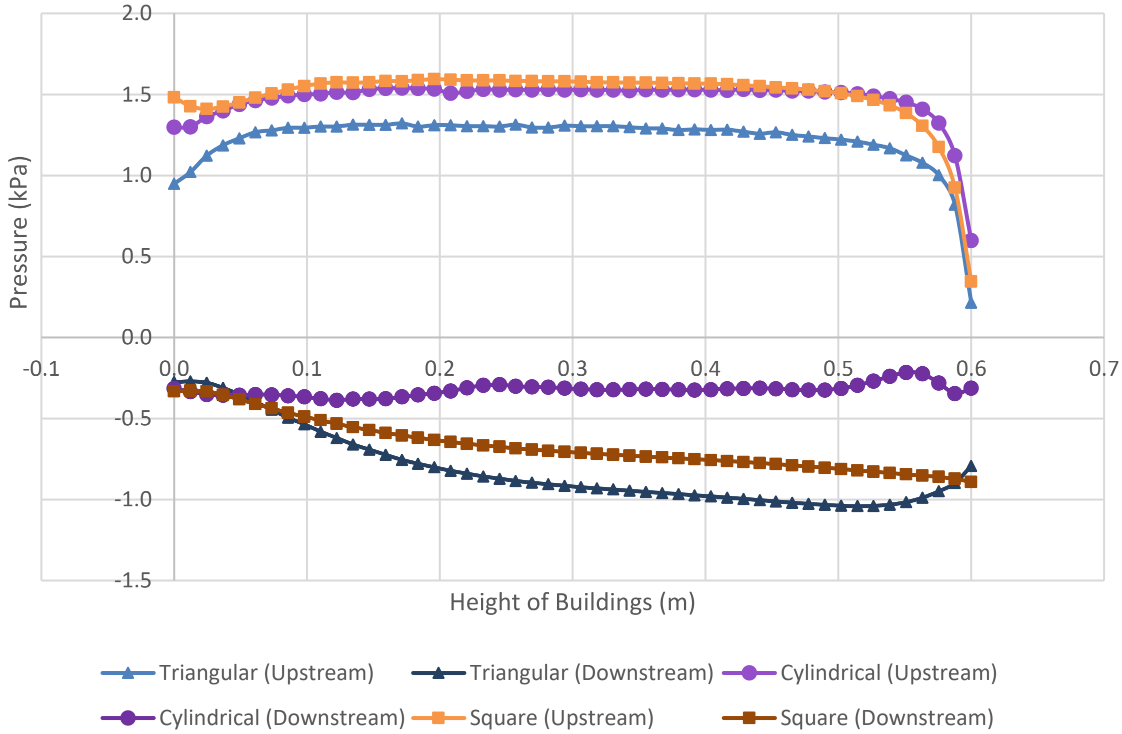


Figure 17. Pressure distribution along the height of the buildings on upstream and downstream.

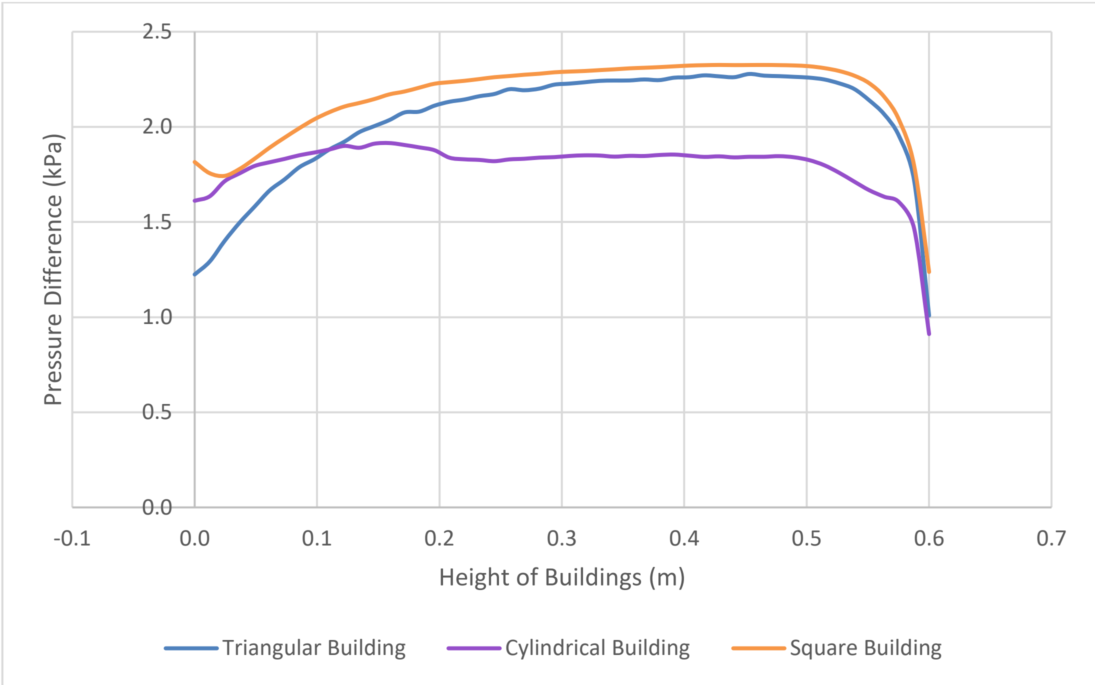


Figure 18. Pressure difference between upstream and downstream of the buildings.

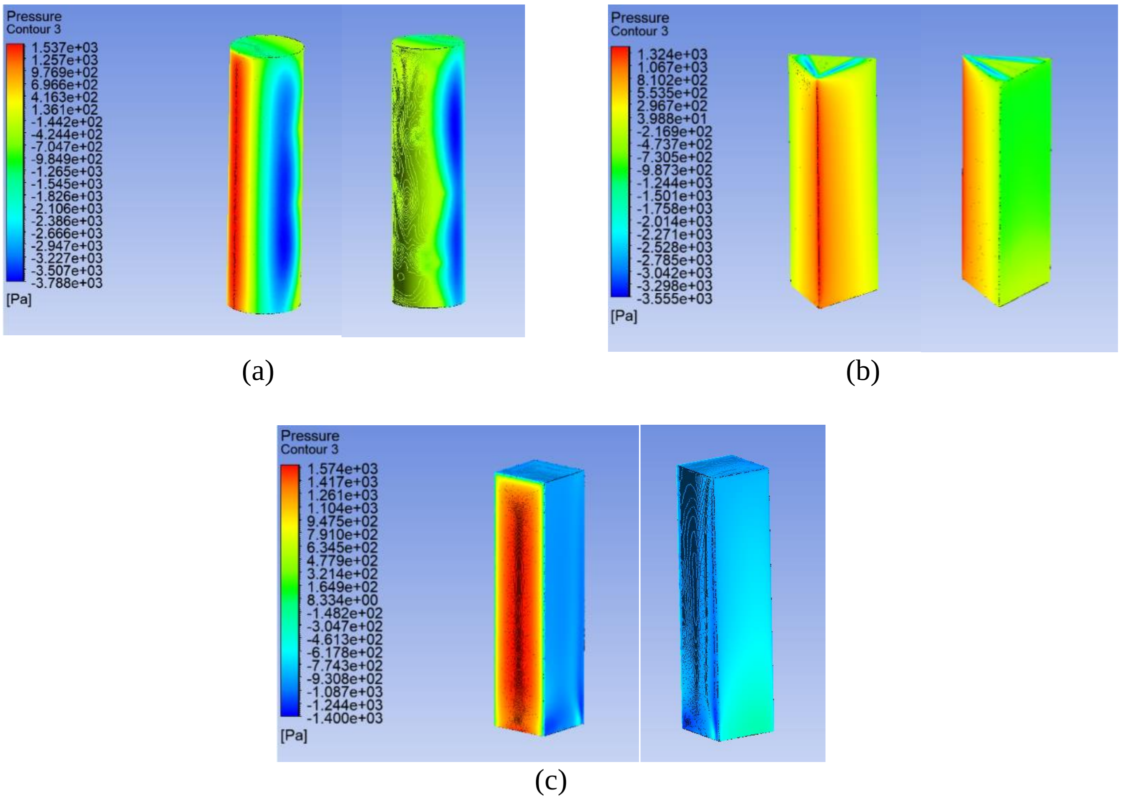


Figure 19. Pressure distribution on different shapes of buildings (a) cylindrical. (b) triangular. (c) square.

4. Conclusion and recommendations

In this research paper, investigations on the effect of wind speed on tall building, effect of height and effect of shape of the building are conducted. The results obtain from these cases are concluded as below.

- The drag coefficient of building is almost constant at low subsonic speed.
- The increase in height of building increases the drag coefficient of the building.
    - The architecture and developer should consider the increment in drag coefficient when planning to increase the height of the building during design stage.
- Drag coefficient of the building is most dependent on the shape of the building compared to the effect of height and wind speed.
- The more streamlined the building is, the lower the drag coefficient of the building.
    - In order to build a taller building, the architecture is recommended to design a more streamlined building such as cylindrical building to minimize the vortex shedding and reduce vibration on the building.

This study provided guidance and recommendations for wind resistance which can be taken into consideration when designing a tall building. Thus, the development of tall buildings can be improved in the future. However, the investigation on the effect of wind direction on tall building is recommended to be conducted in the future research.

One shortcoming of the research is that the wind speed is considered constant along the height of building which is unrealistic. Thus, the velocity distribution is recommended to be applied on the building in future work. Besides, an investigation on effect of interference such as surrounded buildings is recommended to be conducted because the building is assumed to be isolated in the present study. Furthermore, experimental method such as wind tunnel test can be conducted to validate the results obtained from CFD simulation.

Acknowledgements

The authors would like to express special thanks of gratitude to Shahrooz Eftekhari for providing guidance during the research process.

References

[1] 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 *Journal of Wind Engineering and Industrial Aerodynamics* **2**, 213-28

[2] 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* **107-108** 179-91

[3] Guo Y L, Kareem A, Ni Y Q and Liao W Y 2012 Performance evaluation of Canton Tower under winds based on full-scale data *Journal of Wind Engineering and Industrial Aerodynamics*, vol. **104-106** 116-28

[4] Xie J 2014 Aerodynamic optimization of super-tall buildings and its effectiveness assessment *Journal of Wind Engineering and Industrial Aerodynamics* **130** 88-98

[5] Elshaer A, Bitsuamlak G and Damatty A El 2017 Enhancing wind performance of tall buildings using corner aerodynamic optimization *Engineering Structures* **136** 133-48

[6] Neethi B and Joby E 2018 Aerodynamic Modifications against Wind Excitation on Tall Buildings-Shape Optimization *International Journal of Engineering Research & Technology (IJERT)* **7**, ISSN: 2278-0181

[7] 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** 541-8

[8] Venkateshwaran K, Kode K and Thiagarajan K B 2020 Numerical aerodynamic study of a typical high-rise building *3rd International Conference on Frontiers in Automobile and Mechanical Engineering (Fame 2020)* **2311** 030025
[9] Germi M S and Kalehsar H E 2021 Numerical investigation of interference effects on the critical wind velocity of tall buildings *Structures* **30** 239-52
[10] Shafii F and Othman M Z 2004 *Country Report: Wind Loading for Sructural Design in Malaysia*
[11] Malaysia. Jabatan Standard 2002 Code of practice on wind loading for building structure *Putrajaya: Department of Standards Malaysia*
[12] Nizamani Z, Thang K C, Haider B, and Shariff M 2018 Wind load effects on high rise buildings in Peninsular Malaysia *IOP Conference Series: Earth and Environmental Science* **140** 012125
[13] Çengel Y A and Cimbala J M 2010 *Fluid mechanics: fundamentals and applications* (New-York: Mcgraw-Hill Higher Education) chapter 11 p 581
[14] "Menara Citibank - The Skyscraper Center," [Online]. Available: http://www.skyscrapercentre.com/building/menara-citibank/1972
[15] Franke J, Hirsch C, Jensen A G, Krüs H W, Schatzmann M, Westbury P S, Miles S D, Wisse J A and Wright N G 2004 Recommendations on the use of CFD in wind engineering *Proc. Int. Conf. Urban Wind Engineering and Building Aerodynamics*
[16] Lai S and Al-Obaidi A Sh M 2016 Effect of Size and Shape of Side Mirrors on the Drag of a Personal Vehicle *International Engineering Research Conference (7th eureca), Taylor's University* p 782
[17] Hoerner S F 1992 Fluid-dynamic drag: theoretical, experimental and statistical information *Hoerner Fluid Dynamics*