Investigation of wind behaviour around high-rise buildings

Norasikin Mat Isa[1,a]**, Nurul Fitriah Nasir**[1,b]**, Azmahani Sadikin**[1,c] **and Jamil Ariff Hairul Bahara**[1]

[1]Energy and Technologies Research Group (EnRG), Department of Thermofluid Engineering, Faculty of Mechanical and Manufacturing Engineering, Universiti Tun Hussein Onn Malaysia (UTHM)

E-mail: sikin@uthm.edu.my[a], fitriah@uthm.edu.my[b], azmah@uthm.edu.my[c]

Abstract. A study on the investigation of wind behaviour around the high-rise buildings is done through an experiment using a wind tunnel and computational fluid dynamics. High-rise buildings refer to buildings or structures that have more than 12 floors. Wind is invisible to the naked eye; thus, it is hard to see and analyse its flow around and over buildings without the use of proper methods, such as the use of wind tunnel and computational fluid dynamics software.The study was conducted on buildings located in Presint 4, Putrajaya, Malaysia which is the Ministry of Rural and Regional Development, Ministry of Information Communications and Culture, Ministry of Urban Wellbeing, Housing and Local Government and the Ministry of Women, Family, and Community by making scaled models of the buildings. The parameters in which this study is conducted on are, four different wind velocities used based on the seasonal monsoons, and wind direction. ANSYS Fluent workbench software is used to compute the simulations in order to achieve the objectives of this study. The data from the computational fluid dynamics are validated with the experiment done through the wind tunnel. From the results obtained through the use of the computation fluid dynamics, this study can identify the characteristics of wind around buildings, including boundary layer of the buildings, separation flow, wake region and etc. Then analyses is conducted on the occurance resulting from the wind that passes the buildings based on the velocity difference between before and after the wind passes the buildings.

1. Introduction

Buildings are built higher and higher nowadays to support the human's needs and desire, and it is also because of limited space in cities that are crowded and filled with other buildings. Many of the buildings in crowded cities cannot be built anywhere else but up. When building high rise buildings, there are a few design standards that must be taken into consideration such as handling wind issues. As the building increases in height, it subjected to stronger winds that can cause the building to sway. It is known that building structures can be demolished, collapsed or even blown off when it is put against high winds such as typhoons and hurricanes [1].

Thus to prevent such an event from occurring, engineers have to conduct tests on models of the building before hand in order to see whether or not the building can withstand strong winds that flow around the building. Wind or air flow is moving fluid that cannot be seen by the naked eye, and this makes it a bit difficult to study its behaviour [2]. However, through the use of wind tunnel, such study is able to be conducted in order to study the effect of wind around objects, where streamlines are created after the wind flow passes the objects. Smoke is used to make the flow visible to the naked

eye. Through numerical analysis using a software called Computational Fluid Dynamics of CFD, engineers are able to predict and simulate the behaviour of winds around buildings using a computer.

1.1. Literature Review

In modern day, buildings are considered to be the life-support machine, as it is required to provide the necessities for the survival of the human race, such as shelter, security and safety. Not only are buildings needed for humans to be safe from the natural elements that can harm us, such as earthquakes, storms and snow, it is also needed to protect humans from fire. When providing a building of character and aesthetic appeal, it is important that the building is easily, economically and well constructed, with the requirements that it can be easily renovated, maintained, as well as has a sustainable form of construction that can be adapted to changing trends and legislation [3]. Wind, also known as the invisible motion of air with respect to the earth's surface, is caused by the variable solar heating of the earth's atmosphere and is initiated by the difference of pressure between points of equal elevations. The effects on the wind due to friction along the ground can be neglected at a sufficiently great height and the relative horizontal motion of air to the surface is said to be in an unaccelerated flow because of the balance of pressure gradient, the Coriolis and the centrifugal force. However, when near to the ground, the earth's surface exerts a horizontal drag force to the moving air in order to retard the flow. This effect however, decreases, as the height of the flow increases above ground, and as mentioned earlier is negligible above the height δ known as the boundary layer of the atmosphere. It is above this height that the frictionless wind balance takes place with the wind flow with gradient wind velocity along the isobars. Thus, it is of direct interest of the civil engineering structures designer that the wind regime is within the boundary layer of atmosphere [4].

According to Varkute & Maurya, 2013, "CFD has become an influential tool in the building industry. It is used to justify the selection of design option. Not only as a justification or confirmation tool. It has emerged as a shaping and moulding tool also on the drawing board where design is refined" [5].

2. Methodolody

2.1 Scale Models

Using the dimensions of the buildings intended for the study provided by the Malaysian Public Works Department located in Putrajaya, a scaled model of 1:1300 is drawn by using SolidWorks. The drawings of the scale model of the buildings are then sent to be 3D printed to produce a solid scaled model of the buildings for the wind tunnel testing. Scale models of the drawing are then simplified to be imported into the ANSYS FLUENT Workbench for the computational fluid dynamics simulation.

2.2 Wind Speed and Direction

According to Malaysia's Metrological Department, there are four seasons in Malaysia that can be distinguished by the wind direction obtained from the recorded wind flow patterns. These seasons are known as southeast monsoon, northeast monsoon, and two shorter periods of inter-monsoon seasons. The southeast monsoon usually occurs in late June till the end of September while Northeast monsoon occurs in early November till March. The inter-monsoon occurs in between the other two monsoons which are from April to May and September to October (Malaysian Meteorological Department, 2013). The Metrological Department has provided data regarding the maximum daily wind speed near Putrajaya area for the year 2014 until April of 2015. From the data, the wind speed and direction is separated in four groups according to the type of monsoon season. The highest velocity of wind for southeast monsoon is 9.81 m/s with the direction of the wind blowing from the west, the northeast monsoon is 16.9 m/s with the wind blowing from the east and the inter-monsoon of April to May is

10.8 m/s with the wind blowing from the west, and lastly the inter-monsoon of September to October is 13.3 m/s with the wind blowing from the south.

2.3 Wind Tunnel Testing

Wind tunnel testing is conducted on the printed 3d models of the buildings. The building is oriented by a turntable which moves the buildings according to the direction of wind. A Pitot tube is placed at the centre of the buildings. 32 points were marked on the Pitot tube to ease in the taking of velocity data from each of the marked positions. The tests are conducted based on the four seasonal monsoons that occur in Malaysia.

2.4 Computational Fluid Dynamics

The Computational Fluid Dynamics software used for the numerical analysis method is the Workbench15.0. In CFD, the process conducting the numerical analysis is divided into three stages, which are the pre-processing, solvers and post-processing. The type of analysis conducted using this software is Fluid Flow (Fluent). CFD has been proven to be beneficial and useful in various industries as it provides a way to analyse data of a system while saving cost and time from doing the actual experiment. The simplified models are imported into the Workbench 15.0 and the simulation is conducted on the buildings based on the seasonal monsoons that occur in Malaysia.

2.5 Validation and Grid Independent Study

A test is conducted on buildings based on the Southwest Monsoon in order to validate the CFD result with the results obtained from the wind tunnel testing. Furthermore, verification also been done in order to determine which k-ε turbulence models to be used. The grid independent study is also conducted to choose the type of mesh to be used for the simulation. From the validation, it is found that the error between the simulation and the wind tunnel testing is about 17%. The k-ε turbulence model used is the Realizable k-ε turbulence model and the mesh type were determined to be Medium with 70000 elements and 14139 nodes.

3. Result and Discussion

Four wind velocities are used as the initial velocity based on the seasonal monsoons and the buildings are oriented in accordance with the direction the wind. The results obtained from the computational fluid dynamics are separated into three different layers or planes, which are 0.00154 m, 0.066 m and 0.127 m as shown in Figure-1.
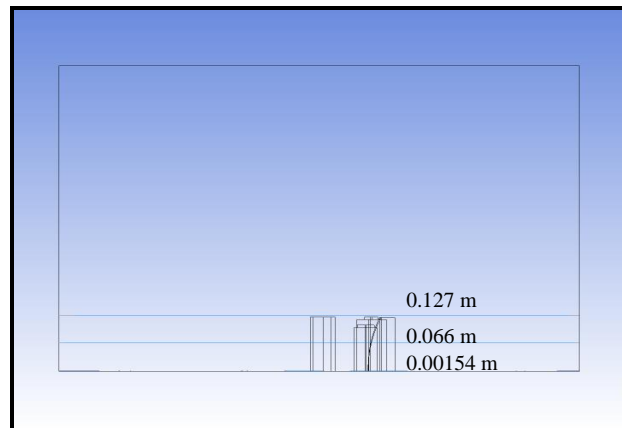
Figure 1: Example of the three planes made for the velocity contour

These reference planes are made to observe the wind behaviour at 2 m from the ground for pedestrian comfort (scale wise), the behaviour in the middle of the buildings and the behaviour that occurs 3 m above the building. From these planes, the velocity contour and the velocity vector can be seen, which represents the wind behaviour around the buildings. The effect of the wind passing the buildings can be seen based on the seasonal monsoons.

The velocity of the wind decreases as it hits the surface of the building due to boundary layer that occurs at the surface. As the wind is stagnated at the surface, it begins to separate, causing the wind to flow around and over the building. This causes wake regions to develop behind the buildings. Recirculation also occurs behind the buildings and irregular vortices mixed together, causing a turbulence behind the buildings. The wind velocity also increases as the separated wind around building mixes together. The wind at the top of the building also increases in flow over the building. An example of the occurring behavior can be seen in Figure-2.
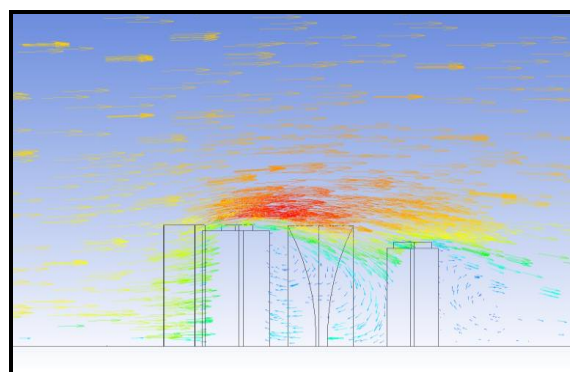

Figure 2: An example of side view of the wind flow behaviour that occurs in one of the Monsoons.

The wind flow velocity contour as seen in Figure-3 shows that, at 0.00154m, the wind velocity is not more than 5 m/s. This means that the wind velocity is suitable for pedestrians, as it is known that only wind velocity of over 20 km/h or 5.56 m/s can bring discomfort to pedestrians [6].
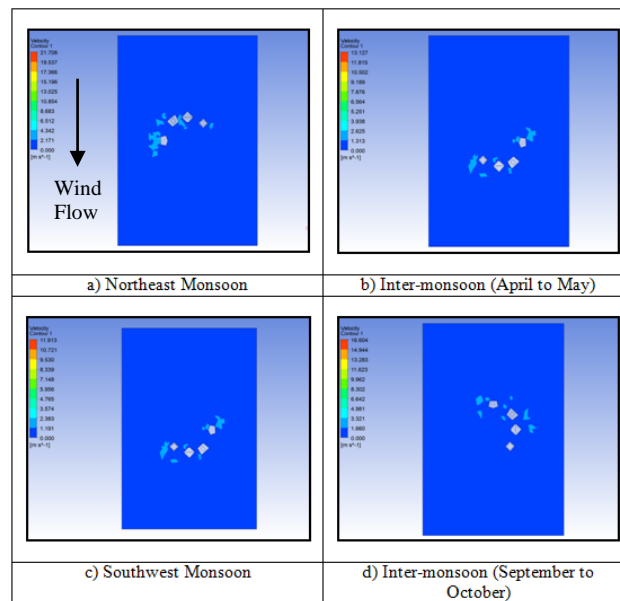
Figure 3: The top view of the velocity Contour of the monsoons at 0.00154 m

The results obtained for all the planes show the different behaviour of the wind as the height of the buildings increase, which can be seen in Figure-4 at the height of 0.06m, and Figure-5 at 0.127 m. Figure-6 shows one of the side views of the velocity contour obtained from one of the monsoons.
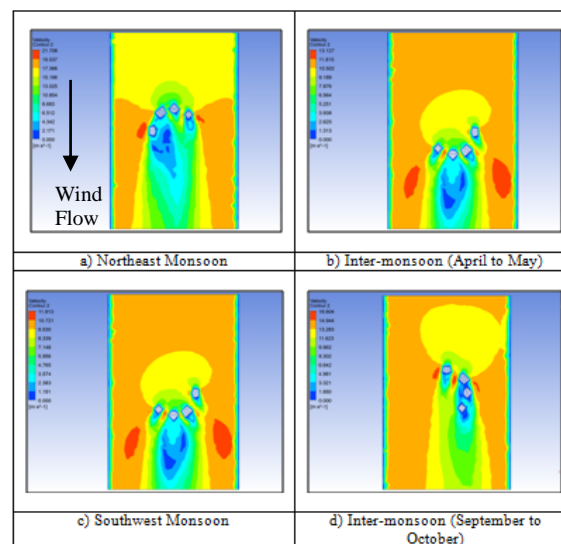

Figure 4: The top view of the velocity contour of the monsoons at 0.06 m

At 0.06 m, the wind flow velocity can be seen changing from its intial velocity as the colour indicates. For the Northeast Monsoon, the initial velocity is 16.9 m/s. However, when the wind hits the surface of the building, the velocity changes to approximately 21 m/s in areas indicated by the red colour. This can be seen with all the seasonal monsoons, such as Inter-monsoon (April to May) with an increase from 10.8 m/s to approximately 13 m/s, Southwest Monsoon from 9.8 m/s to approximately 11 m/s, and Inter-monsoon (Sept to Oct) with an increase from 13.3 m/s to approximately 16 m/s. This increase in velocity is indicated by the red colour as depicted in Figure-6.

The wake region generated behind the building can also be observed, which is indicated by the blue colour, that proves that the velocity is very low – close to 0 m/s – in that area.
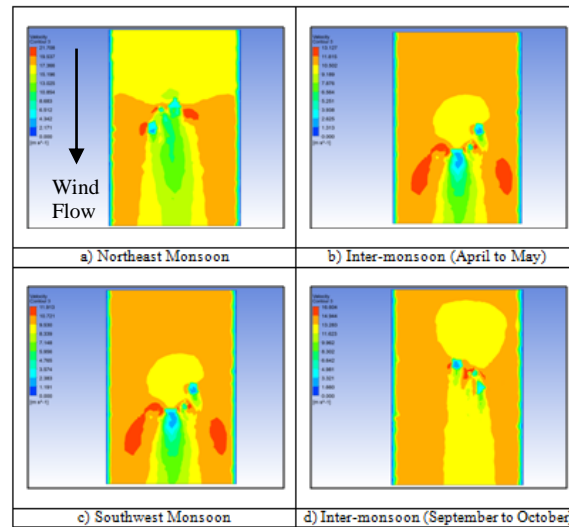

Figure 5: The top view of the velocity contour of the monsoons at 0.127 m

At 0.127m, the wind behaviour that takes place just 3m above the height of the tallest buildings can be seen. The only difference between the wind flow characteristics at this level when compared to that of at 0.06 m is the size of the wake region which decreases and becomes more slender, almost converging back into the uniform flow.
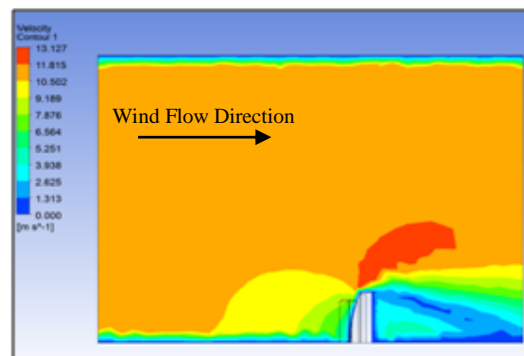

Figure 6: An example of the side view of the velocity contour obtained from one of the monsoons.

In Figure-6, the wind can be seen moving in a uniform flow as it heads towards the building as indicated by the orange colour. As it nears the building, the velocity decreases and becomes stagnated at the surfaces of the building which is caused by the surface boundary layer of the buildings. This forces the flow to separate, which results in the forming of the wake region behind the buildings. The velocity of the wind flow increases as it hit the top edge of the building. This might have been due to the mixing of the separation flow with the uniform flow that occurs above the buildings.

4. Conclusion

From this study, the overall conclusion that can be made is that the flow characteristic and behaviour of wind around high-rise buildings can be seen through the use of the ANSYS Fluent software. There are three focus points that are taken from the y-axis, which are at 0.00154 m, 0.066 m, and 0.127 m. These points represent the pedestrian height of 2 m, half the height of the total buildings, and 3m above the buildings. From these points, the flow behaviour of the wind can be seen through the velocity vector and velocity contour of the wind that hits the buildings based on the seasonal monsoons. There are a lot of advantages when conducting simulation through this software in terms of time and cost. Based on the results obtained from the experiment and the simulation done, there are some differences between them, and that both ways can be used to analyse wind behaviour around buildings. The differences that were obtained might have been caused by the simplified models of the buildings used in the CFD. Through the CFD simulation, boundary layer, wake region, separation of flow, and recirculation of flow can be seen. The layout of the building also plays an important role in reducing the negative effect of winds, such as gust and whirlwind. Thus, the distance between the buildings must be taken into consideration when building high-rise buildings. The results that were obtained also show that the wind that hits the building is below the discomfort velocity. This proves that a lot of aspects have to be taken into consideration when building these high-rise buildings.

5. Acknowledgement

References

[1] Holmes, J. D., 2007. *Wind Loading of Structures.* 2 ed. s.l.:Taylor and Francis Group.
      Malaysian Meteorological Department, 2013. *Official Website Malaysian Meteorological Department.* [Online] Available at: http://www.met.gov.my/index [Accessed 17 2 2015].
[2] Sato, M. & Takaya, K., 2012. A fundamental study of the flow past a circular. *SIMULIA Community Conference,* pp. 1-5.
[3] Osbourn, D. & Greeno, R., 2007. *Mitchell's Introduction to Building 4th Edition.* 4 ed. s.l.:Pearson Education Limited.
[4] Simiu, E. & Scanlan, R. H., 1996. *Wind Effects on Structures.* s.l.:A Wiley-Interscience Publication
[5] Varkute, N. S. & Maurya, R., 2013. CFD Simulation in Township Planning- A Case Study. *International Journal of Computational Engineering Research,* 3(3).
[6] Planning and Building Department, Development and Design Division, City of Mississauga, June 2014. Urban Design Terms of Reference. *Pedestrian Comfort and Safety Studies.*