Analyzing the Ventilation Performance of Tropical High Density Residential Precincts using Computational Fluid Dynamics

HII JUN CHUNG Daniel, CHYE KIANG Heng, LAI CHOO Malone-Lee and JI Zhang
Centre for Sustainable Asian Cities, National University of Singapore
sdedhjc@nus.edu.sg, sdedean@nus.edu.sg, sdemalon@nus.edu.sg, sdezj@nus.edu.sg

Abstract. In high rise, high density tropical cities, we expect the urban fabric to heavily alter the natural wind pattern of the city. The thermal comfort of the inhabitants will be affected if the desired ventilation performance is not achieved. In addition, the accumulation of pollutants and heat will affect the quality of life in the city, causing health problems and also resulting in greater energy consumption for cooling. This exploratory paper presents a comparison of four tropical high density residential precincts to analyze their natural ventilation performance using the Computational Fluid Dynamics approach. The methodological tools used in the process systematically demonstrate the work process flow and outcomes that can be achieved. In addition, the paper explores how collaborative design can take place in the CFD work environment to cover different typologies and wind directions. The investigation is helpful to show how CFD simulation can be more closely integrated with the design process and enabling more design firms to use it in their increasingly larger projects.

1. Introduction

With increasing rate of urbanization, cities in the world will continue to experience large population increases, and many high density cities like Singapore and Hong Kong have to be built to even higher densities. Where urban land is a constraint, high-rise and compact urban forms become inevitable solutions. In such situations, the natural wind flow patterns in the local areas can be significant altered. This interaction between the urban areas and the...
atmospheric conditions has received much attention in urban climate research in the last decade [1, 2, 3]. Alteration of the micro-climate caused by rough urban forms is generally not desirable since wind flow helps to maintain the thermal comfort of the residents as heat and pollutants are being channeled out of the city, and help mitigate the Urban Heat Island effect [4]. In the tropical context, wind flow is particularly crucial to maintain people's thermal comfort given that the ambient air temperature is generally high from the full exposure of the sun for almost the entire year. Other climatic factors affecting thermal comfort apart from wind speed are temperature, relative humidity and mean radiant temperature. Studies have shown that wind flow plays the most significant part in maintaining human comfort despite under full exposure to direct sunlight. Hence, wind flow analysis is crucial in sustainability studies to attain the objective of making the building or precinct design more energy efficient through reducing people’s dependence on mechanical ventilation to compensate for inadequate wind flow.

Computational Fluid Dynamics (CFD) has been increasingly used in the field of architecture, urban design and urban planning to understand the patterns of wind flow through the built environment, apart from traditional wind tunnel tests and on-site measurements. It is one of the various disciplines involved in urban climatology [5]. The availability of more powerful hardware for mainstream computer users as well as the lowered costs of these computers made CFD more accessible for adoption in the design professions today. It is also a faster, cheaper and easier approach compared to traditional methods. This means greater integration of CFD into the design process, especially to analyze the impact of the design on the current site conditions and annual wind patterns will help make the new designs more responsive to the natural ventilation on site. For today's larger scale of design project development, the collaborative design approach is also important to integrate the entire team into the workflow processes. This approach saves time and costs as repetitive work is avoided.

The interest of this paper is to analyze the high density building typologies to comparatively analyze their respective responses to the local wind flow pattern. A typology may be considered desirable when the wind flow can penetrate the site with minimal obstructions, and also in the process, it does not create stagnant spaces, generating vortices and turbulences. As the study aims at comparative analysis of various local high density typologies, the discussion can assist in planners and architects in identifying the most promising typologies in terms of their response to the wind, and further study the contributing design factors. The discussion also alludes to how CFD simulation can be executed in the collaborative design manner for greater work efficiency.
2. Related works

There has been extensive research done in terms of form, morphology and typology studies and how they affect natural ventilation. However, most of the research is on generic forms and idealized arrangements of the urban geometries. Many of these studies, apart from providing better understanding on wind pattern affected by urban form, also look at how they affect heat and pollutant dispersal. They are mostly done in the CFD domain with validation from wind tunnel tests, which have shown general agreement with the simulation results [6]. There has also been research [7] which integrates form generation with CFD analysis results.

Another interesting study explored the terms "city breathability" and "pollutant dilution" within several urban forms [8] by investigating the mean age of air. It explored the behaviour of the wind flow by categorizing the city into three forms: sparse, compact and very compact forms. The impact of the appearance of recirculation zones in the different urban forms show how breathable a city is – if there is sufficient wind flow through the city. Another study on high-rise compact urban areas was done to identify obstacles and pathways to the approaching wind [9], and wider streets and smaller building area density were found to provide more wind pathways and larger flow rates along street networks. As flow rates along the street may quickly decrease due to strong resistances by high-rise buildings, the suggested design response was that total street length should be limited or ventilation in downstream regions will be badly curtailed. Secondary streets always suffer worse ventilation outcomes than the main streets, and this problem can be mitigated by having variation of building heights.

In terms of city level exploration, a study looked at the relationship between pollutant dispersion and urban morphology [10]. Three city forms are used, namely sharp-edged round city model, smooth-edged round city model and sharp-edged square city model. The overall city form, the configuration of streets, the upstream wind directions are all shown to be crucial urban morphological parameters in affecting urban pollutant dispersal.

Studies have also shown that having a variation of building height will ameliorate the UHI effect, for example, in the case of Singapore [11], these include introducing high-rise towers at intervals, maintaining height-to-width ratio of 0.6 to 0.7 to give maximum velocity at the center of the urban canyon as well as testing parallel and perpendicular flow of wind against the canyon. This is especially effective when the wind flow is parallel and perpendicular to the canyon [12]. Another study demonstrates that better overall performance could be obtained by varying the skylines which improve around 20-30% for daylight and 35-70% for air ventilation [13].

In a study in Hong Kong, two performance indicators; velocity ratio and retention time of pollutants at the street level, are used to quantify the air
ventilation impacts [14]. Results show that velocity ratio at 2m above ground was reduced by 40% and retention time of pollutants increased 80% inside the street canyon as high-rise buildings with 4 times height of the street canyon were aligned as a wall upstream. The recommendation is to avoid such shield effect to allow the wind velocity to channel through the city.

A study is done on the effect of building grouping pattern on the wind environment in the outdoor spaces and the resulting ventilation potential of these buildings [15]. It has been found that grouping pattern of buildings as well as their orientation with respect to wind has a dramatic effect on the resulting airflow behaviour and pressure fields. Configurations that contain a central space articulated by buildings and oriented towards the prevailing wind can offer better exposure to air currents and better containment of wind. Apart from thermal comfort, minimizing energy consumption is another crucial factor for more sustainable cities, and a study was conducted to explore forms which minimize solar energy falling on roofs and on the ground of surrounding buildings and increasing airflow between buildings [16]. As a recommendation, the study proposed a typology named Residential Solar Block (RSB) which has a form that aims at achieving energy efficiency and is far more superior than the linear urban form and block urban forms used for comparison in the study.

3. Method

This research paper links Rhinoceros 4.0, a CAD software with ANSYS Workbench 12.1, an integrated platform of various analysis systems, including fluid flow. The user can visualize the results in the virtual reality (VR) environment in CEI Ensight at the end. The entire workflow is shown in Figure 1. This study uses FLUENT 12.1 under the Workbench platform for CFD wind flow simulation. The more robust and accurate 3D CAD format, STEP.stp is exported from Rhinoceros to be imported into the Workbench platform.

In the Workbench platform, there are three stages – pre-processing (DesignModeler and Meshing), processing (FLUENT CFD) and post-processing (CFD Post). The Workbench platform is also the location where collaborative design can take place, as this is where most of the design exploration and different parameter outcomes analyses are possible. This is because of the nature of the procedural process in Workbench where interventions can happen at any level of the entire workflow.
The urban environment to be simulated in the study has two predominant wind directions of north-east and south-west, which are basically the two major monsoon seasons of the year. The annual average wind speed used is 2.9 m/s (north-east) and 2.3 m/s (south-west). The domain for the wind flow simulation must be at least 3 times the size of the characteristic lengths of the precinct, up to 5 to 10 times as shown in Figure 2. A cylinder is chosen as the domain shape for the wind flow simulation as it is easier to change the wind direction using this. Ideally, the facades should align to both north and south directions to take full advantage of the predominant wind directions. This is actually a good practice since avoiding any major facades facing the east and west directions will overcome direct sunlight exposure.

As for meshing, the mesh density should be in the range of 0.5 to 1 m near buildings and at ground level to 10 m at far field as shown in Figure 3 [17]. This will give a more detail visualization of the wind flow around the precinct and far less attention to the cylinder’s periphery. Tetrahedral meshing is used in this study with the commonly used realizable K-epsilon turbulent model. The turbulent viscosity model is obtained by solving two additional transport equations. Basically it solves transport equations for the turbulent energy, \( k \) and its dissipation, \( \varepsilon \) [18].
Fig. 3. The precinct with finer tetrahedral meshing at the centre to a gradual coarser meshing to the edge of the cylinder.

In the Workbench platform itself, because of its procedural workflow nature, the opportunity to do collaborative design is possible. This is important as there is a requirement for ventilation assessment of the design projects nowadays to demonstrate the ventilation performance from all wind directions. Therefore, the tasks can be broken down to parts to be done by the team members in parallel. As illustrated in Figure 4, a precinct for wind flow simulation can be divided into 8 parts for 8 different wind direction analyses. 8 different technicians can connect to a single 3D model and be responsible to simulate the wind flow analysis of each wind direction. Another technician can take results from each wind direction and compare them side by side for further analysis. Further, another team of professionals responsible for another design can bring their design to the table and compare with the first team’s design. Finally, parametric studies can be done for every precinct to explore the different ventilation performance by changing different parameters like height and orientation. These combinations are endless so the scale of collaboration can be as broad as possible.
4. Results

Four local high density typologies at urban precinct levels are used for this comparison study. All of them have multi-storey car parks at 4 storeys high. Figure 5 shows the plan and elevation of the precincts.

These precincts place the multi-storey car park at the centre to the corners of the site to achieve different configurations. From initial plan observation, Precinct
01 is the only typology that opens up clear paths aligning to the predominant north-east and south-west wind directions as well as having higher permeability into the precinct. On the other hand, Precinct03 will be the opposite in that it is shielding the courtyard space heavily. The only potential wind directions to enter the courtyard are the north-west and south-east directions, which are rare. This implies that it is expected to perform the worst among the four precincts. These observations were validated in the simulation exercise. All precincts have blocks around 16-18 storeys of height so the factor of height is not a major issue that will affect ventilation performance.

For the north-east wind direction as shown in Figure 6, we may summarize that putting the lower multi-storey car park at the corner is considered good only when the wind can blow directly over it in the same manner as in Precinct01.

Although the north-eastern block shields almost 50% of the site from the north-east wind, there is at least a clear opening for cross ventilation through the precinct. Precinct02 has very thick point blocks which shield majority of the wind from entering the centre of the precinct although there is a very small gap.
Precinct03 is the worst precinct in terms of getting the wind flow through the site with heavy shielding of the prevailing wind. Most of the wind flow has to go in from the lower multi-storey car park at the sides. Precinct04 is the best typology for the two predominant wind directions as there is a generous gap for both wind directions to enter the courtyard.

Figure 7 shows the north-east wind velocity vector on isometric view. It actually gives a very good indicator of how the wind behaves in 3D, showing clearly the blue zones where obstructions occur and create turbulence. It is not surprising to see the large amount of blue zones in Precinct03 just like in the plan view.

For the south-west wind direction as shown in Figure 8, there is apparently no problem of cross ventilation in Precinct01 as most of the blocks are aligned to allow wind to enter the precinct except for the north-eastern block which is being shielded once again. For Precinct02, the performance is slightly better than the north-east wind as there is a huge gap that allows the wind to channel through.

For both wind directions, Precinct04 has the least chances of generating turbulence and vortices as there are less convoluted facades facing the wind.
directions. All the rest have that issue, especially Precinct03. Turbulences and vortices are not ideal occurrences in the built environment as apart from creating dead spaces, they are also slowing down the wind velocity entering a precinct, which in the case of Precinct03 case has caused the centre of the precinct to be lacking of wind velocity vectors.

Fig. 8. The south-west wind velocity vector on plan view.
Figure 9 shows the south-west wind velocity vector in isometric view. Just like the plan view and identical to Figure 7, it shows a lot of blue zones, which means a lot of turbulence occurring in the obstructed areas. This is definitely not ideal for cross ventilation through the site as it acts like a wall or barrier to block off potential wind flow.

The above visual observations demonstrate that precinct typology can be more sensitively configured to allow more wind path for both the north-east and south-west wind to order for the cross ventilation to occur in the precinct. Precinct03 is having the most obstructions facing the predominant wind directions. The observation would point to the need to minimize direct obstructions in typology design by different orientations and arrangements in the context of tropical hot and humid regions as it will lower down the wind speed. The results are only preliminary findings and further comparative analysis need
be done after this by taking into account other issues to be discussed later in future works below.

Table 1 shows each precinct’s average velocity magnitude for both the NE (North East) and SW (South West) wind directions but taking the elevations of pedestrian (2m) and roof garden of the multi-storey carpark (14m). From the results, it is evident that Precinct004 is the overall best performer by looking at these two levels with higher average wind speed. This can be attributed to the generous spacing between the blocks as well as the clear openings for both predominant wind directions to enter the precinct.

Table 1. Each precinct’s average velocity at 2m and 14m elevations.

<table>
<thead>
<tr>
<th>TYPOLOGY</th>
<th>2M AVERAGE VELOCITY (M/S) NE</th>
<th>14M AVERAGE VELOCITY (M/S) NE</th>
<th>2M AVERAGE VELOCITY (M/S) SW</th>
<th>14M AVERAGE VELOCITY (M/S) SW</th>
</tr>
</thead>
<tbody>
<tr>
<td>Precinct001</td>
<td>0.428</td>
<td>1.267</td>
<td>0.336</td>
<td>0.988</td>
</tr>
<tr>
<td>Precinct002</td>
<td>0.417</td>
<td>1.134</td>
<td>0.340</td>
<td>0.915</td>
</tr>
<tr>
<td>Precinct003</td>
<td>0.382</td>
<td>1.003</td>
<td>0.311</td>
<td>0.794</td>
</tr>
<tr>
<td>Precinct004</td>
<td>0.501</td>
<td>1.596</td>
<td>0.393</td>
<td>1.262</td>
</tr>
</tbody>
</table>

5. Conclusion and future works

This paper is an initial exploration of precinct typology comparison in wind studies using CFD. It mainly tests the methodological tools and approach to the particular simulation exercise. The results are visual and indicative, and more detailed analysis would be necessary using other matrices.

Our future work will include looking at relationships between wind flow geometric variables and ventilation performance indicators. There are many ventilation performance indicators that could be explored which include the pressure coefficient, drag coefficient and wind velocity ratio. As for geometric variables, they include frontal area density, building height to width ratio and permeability. The pressure coefficient is a good indicator to detect potential cross ventilation through the typology when there is a difference in pressure distribution. The drag coefficient can quantify the resistance of the built environment towards the wind. The wind velocity ratio [19, 20, 21] is a measurement of how much wind velocity actually made it to the streets and podiums in the outdoor environment. It is a ratio of the wind velocity at the street level over the wind velocity above the urban fabric. The frontal area density [22], which is used to indicate the roughness of the urban fabric and to identify corridor
paths, could also be a good geometric indicator for each precinct to understand
the average façade areas that are blocking the wind from every direction. Overall,
it may be necessary to selectively find the relationships as this could give a quick
estimation of the ventilation performance of a design before conducting CFD
simulations. This will be helpful to aid the initial conceptual design process of the
architects and urban designers.

In terms of further research, other turbulent models such as large or detached
eddy simulation (LES or DES) could be employed for more accurate wind flow
simulation, but it will be far more time consuming. Apart from that, integrating
the wind flow and thermal simulation could potentially present a better
understanding of the overall picture of how heat transfer occurs in the urban
environment, as the direct radiation from the sun causes atmosphere temperature
differences which in turn give rise to pressure differences that affect natural
ventilation.

In addition, in comparing typologies, it may be necessary to decide on the
extent to which crucial building element features should be considered when
doing typology studies. These may include double volume spaces, bridges, void
decks, buildings on columns, roof profiles, balconies, air-conditioning
condensers, façade materials and even the presence of vegetation in the precincts.
All these potentially have effect on how wind flow will behave in the precinct,
and it may be necessary to consider how to standardize all these features before
comparisons can be made across the board.

Finally, a normalization process [23] has to be in place to surround the precinct
of interest with identical copies of similar density and height, as shown in
Figure 10. The purpose is to control the uneven impacts of the surroundings of the
real site so a theoretical homogenous context to evaluate the theoretical
ventilation performance of a precinct can be done. This normalization process
could include adding more identical copies in both X and Y axes to a stage where
the wind flow pattern is consistent before we take the wind velocity readings of
the precinct of interest in the center. This could be done for 8 orientations so we
can assess its true average performance as well as the best and worst orientations
for the local wind pattern.
Fig. 10. The precinct of interest in the centre to be surrounded by normalized identical copies to create a fairer context of similar height and density.

In conclusion, given that designing in response to wind is extremely important in tropical dense environments; various methodological tools should be explored to draw the appropriate relationships that can aid design and planning decisions. The paper is doing preliminary explorations of typology ventilation performance comparison using CFD and its further use can be refined through more detailed studies with support from other matrices.

6. **Acknowledgement**

This paper is part of an on-going research that is funded by the Singapore’ Ministry of National Development Research Fund with the participations of the Urban Redevelopment Authority and the Housing Development Board. We acknowledge the information provided for the paper is derived from these government agencies that are collaborating in the project. The paper should not be quoted without authors’ permission.

**References**

ANALYSING THE VENTILATION PERFORMANCE OF TROPICAL HIGH …

D. HII JUN CHUNG, H. CHYE KIANG, M.-L. LAI CHOO, Z. JI

