Use of CFD for Wind Flow Analysis in Urban Areas

Daniel Hii Jun Chung, Centre for Sustainable Asian Cities, School of Design and Environment

Background

Singapore is a city state with limited land for development. With population and economic pressures, the built environment today consists primarily of high rise and density developments. Such built forms inevitably affect the natural wind flow patterns. Given that the average annual wind speed is quite low at around 3m/s, with no wind experienced for about 25% of the year, it is critical to design our built environment without negatively impacting on the already low wind speed conditions. In the present hot and humid environment that we are living in, the two best passive methods to cool down the built environment is to make use of natural ventilation and greenery. These not only provide thermal comfort to residents but also cut down on energy usage in buildings, especially from cooling load demands. Apart from heat dispersion, pollutant dispersion is also important to keep the city healthy for outdoor activities. Using ANSYS Fluent, we can harness the capabilities of the Computational Fluid Dynamics (CFD) simulation to help us predict how much of the wind flow is altered by building designs.

Demonstrating the use of CFD for Wind Flow Analysis

The use of CFD simulation for planning was demonstrated in a local area situated at the south-eastern part of Singapore around the Kallang River where the conceptual design with massing models is publicly available. This is one of the four growth areas identified in the Singapore Master Plan 2008, a statutory plan that shows the development patterns envisaged for a projected population of over 5.5 million in the future. It is an inner urban area with a site area of 64 hectares (640,000m²) of prime developable land. The proposed Plot Ratio (Floor Space Ratio) for the site ranges from 1.5 to 5.6, which will have building heights from three to 36 stories. CFD simulation can quickly tell us areas in the whole zone where the air circulation is bad (highlighted by the red box) for both the predominant wind directions (north-east and south-east).
Path lines coloured by Velocity Magnitude (m/s) of the Kallang River north-east wind flow analysis in ANSYS Fluent. The colour of the lines ranges from red (higher velocity) to blue (lower velocity)

Path lines coloured by Velocity Magnitude (m/s) of the Kallang River south-east wind flow analysis in ANSYS Fluent. The colour of the lines ranges from red (higher velocity) to blue (lower velocity)

Animated path lines can be viewed [Visualisation Showcase](#) at HPC webpage or the following URLs:-

- [mms://live-vip-49.nus.edu.sg/all_publicEvent/WebcastEventVideo/Yr2010/kr_topview_512k.wmv](#)
- [mms://live-vip-49.nus.edu.sg/all_publicEvent/WebcastEventVideo/Yr2010/kr_frontview_512k.wmv](#)
- [mms://live-vip-49.nus.edu.sg/all_publicEvent/WebcastEventVideo/Yr2010/kr_birdseyeview_512k.wmv](#)

**Studying the Ventilation Performance of Precinct Level Residential Typologies**

Wind, like solar power, is a directional force that affects our built environment. Urban morphological and geometrical variables do affect the wind flow in urban areas, such that the overall wind velocity is altered. As we plan for the future housing of Singapore, we can collect residential projects in high density precincts
from around the world to test their ventilation performance. By collecting a large number of case studies, we can learn how variations of building shape, geometry, orientation, site coverage, façade area, height, arrangement and permeability affect ventilation performance. We can measure the velocity magnitude as well as the age of air in order to rank them at the 2m pedestrian level as well as the 2/3 height where the stagnation zone appears.

The following are factors that can affect ventilation performance:

- **Building shape**: The location of sharp edges relative to each other affects the vortex structures at the wake areas where the wind flow goes over or above buildings. Building roof shape profiles can determine whether wind flow can enter the spaces between buildings. Some building shapes will retain or deflect wind flow more while others allow wind to flow through them.

- **Geometry**: Wind flow will decrease when the canyons are deeper and longer. This means that, as a general rule, we should never build long slabs that shield wind from entering and result in a long time for wind to exit.

- **Orientation**: Buildings need to be orientated to allow the predominant wind directions to flow through the entire city. It is not good to create too many obstacles for the wind to be enjoyed by some while the rest of the population enjoys nothing.

- **Site coverage and façade area**: Generally, the higher the site coverage, the more obstacles of façades blocking the predominant wind directions. This will slow and block wind from flowing through the city.

- **Height**: The variations of height is usually good for wind flow to enter the spaces between buildings to ventilate them. If all buildings are of uniform height, wind does not have the chance to ventilate the spaces between buildings.

- **Arrangement**: As a rule, buildings should always be arranged to give clear passage for the wind to flow through the city. When there are limited choices available for direct clear openings, staggering the buildings will help the wind flow to permeate through the urban fabric indirectly.

- **Permeability**: When there is not much that can be done in terms of building orientation, arrangement and site coverage, the only solution is façade obstruction intervention. Double volume spaces, cross ventilated corridors and void decks are some building design features that can help improve wind flow. This means the façade obstruction is minimised by introducing openings or holes in the building facades to allow wind to penetrate or permeate through the obstructions.
Using HPC’s Parallel Processing

For wind studies relating to precincts, a precinct size may range from 45m to 150m in length and width while the height can go as high as 138m. By putting the typology into the domain for simulation, the dimension can be as much as 1,200m to allow enough space to eliminate interactions between flow disturbances and boundaries as well as for turbulent dissipation. In our experience with studying housing typologies in international case studies, the range of samples could result in mesh sizes ranging from 2.29 million to over 23.75 million elements, with the minimum mesh dimension of 0.5m and growth rate of 1.1. Normally, typologies with many convolutions will give us a lot of mesh count, which is quite common in new designs that are no longer simple straight or linear blocks. This could give us a Fluent .cas and .dat file size of 3.4GB each. If we were to simulate them based on four to eight of the most predominant wind directions we face in Singapore, the computing power required would be tremendous. The HPC’s parallel processing capabilities can help speed up the time taken to simulate such large cases, especially those with mesh sizes beyond 10 million elements.

An example a rectilinear typology (N Joy, Zurich - above) and an example of a very convoluted typology (King Lam, Hong Kong - below) which plays a significant role in their total meshes amount difference.

In our studies, the use of Realizable K-epsilon turbulent model was selected because it is among the best
steady-state models that can model the separation flows well, especially for urban level scales. We need to run them at second order scheme for all parameters to give us acceptable results as well as residuals convergence with the termination criterion at 0.0001.

**Conclusion**

In conclusion, CFD benefits our research by cutting down the time and costs spent in parametric, case and design exploration studies. Wind tunnel tests are important for validation but performing the hundreds of simulations needed would be too costly and time consuming. In recent years, better integration with Computer Aided Architectural Design (CAAD) formats has benefitted the architectural, building and construction industry. ANSYS Workbench has made it easier for us to import our 3D models for the CFD simulation. Many formats can be exported directly via Rhino, Revit, AutoCAD and 3ds Max. The HPC parallel processing allows us to run very large and complicated cases. It is more logical to run larger projects with at least 16 cores and beyond. Powerful workstations today are only capable of running eight cores at one go per CPU (or course, we can expand to two CPUs for 16 cores) with the limitations of 48GB RAM. This means meshing large projects will have hardware constraints while running the CFD simulation is very time consuming. Using the GPU’s capabilities is still not in the realm of commercial CFD software and running commercial CFD software on cloud computing is not possible now.

For our future plans, we may need to run very large sites for design exploration exercises. Such sites may range from 2.5kmsq to 8kmsq and beyond to better understand their current conditions. Apart from that, running transient turbulent models like Detached Eddy Simulation and even Large Eddy Simulation will provide us with more accurate results. Both these options are possible if we can harness the parallel processing power of the CPU beyond 16 cores that we are using now. Perhaps we can even harness the GPU parallel processing if it becomes easier to adopt into the workflow with the promise of more affordability per core.