COMPUTATIONAL FLUID DYNAMICS FOR URBAN DESIGN

DANIEL HII JUN CHUNG and LAI CHOO MALONE-LEE
Centre for Sustainable Asian Cities, School of Design and Environment, National University of Singapore
sdedhjc@nus.edu.sg, sdemalon@nus.edu.sg

Abstract. Computational fluid dynamics (CFD) is a method of solving and analysing problems that involved fluid flows. In the field of architecture, urban design and urban planning, CFD is useful for the analysis of ventilation and airflow in the built environment, especially in very dense cities. This paper will look into the possibility of making CFD more accessible to the general design and planning field. A simulation is done on a urban design proposal to quickly see how air flow behaves around it. From there, it looks into the future where technology will make CFD simulation more easily adopted and the possibilities of integrating the ventilation analysis with other environmental analysis results into the urban design arena.

Keywords. Computational fluid dynamics; sustainability; high density; urban design; airflow.

1. Introduction

Today, compact and high-density environments are often advocated to accommodate more people in urban cores to prevent urban sprawl as well as wasteful use of land and natural resources. However, such environments do generate high levels of heat and consequently require more energy for artificial lighting and air-conditioning to make the spaces conducive and comfortable for work as well as living. Therefore, efficient urban design and planning is required to overcome these concerns.

Traditionally, land use planning addresses urban functional and intensity issues through two-dimensional zoning approaches. In dense urban areas where urban form and cityscapes are important concerns, such land uses are
simulated in urban designs and then translated to planning guidelines that establish the relationship of form, mass, space and function, particularly as they affect the quality of living environment for inhabitants. However, issues such as air stagnation and airflow in the immediate context are usually not analysed at this stage, and very often, this is left to designers at the architecture/building design stage to assess the need for ventilation design interventions at the final stage, particularly where this has an impact on the assessment rating for sustainable designs. We propose that CFD can be done at the urban design stage at a bigger scale as shown in figure 1.

Figure 1. Planning and design process.

Today, sustainability has become a major concern in urban areas as energy consumption, excessive waste and pollution are threatening the resource equilibrium of the world. Issues such as urban heat islands, overcrowding, traffic congestion, uncomfortable pedestrian spaces and inadequate open spaces are resulting in deteriorating living and working conditions which affect human health and wellbeing.

One of the ways to mitigate high energy consumption is to plan and design urban spaces with due regard to existing ventilation patterns and heat concentration of the specific urban area. This can be done by appropriate studies at the urban design stage, using computational fluid dynamics (CFD), a technique pioneered by Harvard Lomax (Lomax et al., 2002). The technique can be used to analyse prevailing wind direction and speed as well as heat radiation patterns. It is particularly useful in high-density and high-rise contexts where the built environment has extensively altered the natural ventilation of the space.

1.1. OBJECTIVE

The aim of this paper is to look at making CFD more accessible to the planning and design practice as well as the architectural, urban planning and design researchers. At this stage, CFD is still beyond the reach for many, and at the university level, it is mostly offered on a shared platform between the academic and research community like our university’s High Performance Computing (HPC). The problem with this situation is that there is usually a contention for the simulation facility, with queuing time taking away valuable research time.

The platform running in the HPC is in both Linux and UNIX mode. Most of the application software that is currently being used in the computer-aided
architectural design (CAAD) world is running on either the Windows or Macintosh platform. This is a major incompatibility issue that makes it difficult to integrate to the workflow. Apart from that, most commands are inputted line by line without a graphical user interface (GUI). This makes it very tedious for designers who are used to operating on GUI mode. Even if the more traditional approach of using wind tunnels is resorted to, investment is required in terms of the space and physical facilities required, which again poses a problem when space and funding resources are constrained.

Therefore, CFD analysis is not commonly done in design, let alone at the earlier planning and urban design stages. Den Hartog et al. (2009) highlighted that CFD will provide architects with insights into the effects of unusual architectural forms on the climate to give clues on potential building services problems. Architects can plan their designs efficiently just by analysing these results, especially in the initial stages, if they can understand the general patterns of the airflow.

Many researchers or practitioners in the field often sub-contract the CFD simulation work to consultants or specialists. However, this usually takes too much time and does not allow for interactive responses given that design is always evolving according to many factors. There have been intentions to integrate it into the design studios but the results were not satisfactory (Tsou, 1998; Tsou et al., 2001a). Today, we are living in an exciting time as the technology has reached a stage where local desktops and workstations are starting to be able to do some degree of complicated simulations which a few years ago remained in the realm of supercomputing. This opens up new opportunities to consider the integration of CFD simulation at the earlier stages of the planning and design processes.

Our main research focuses on the density thresholds of urban environments and their corresponding environmental factors. As mentioned, CFD is one of the parameters we look into to assess good ventilation for the urban-scape. In other words, we propose to improve the urban design parametrically, with regard to building intensities, heights, forms and other morphological attributes, based on the environmental analysis inputs. Malkawi et al. (2005) had done similar work in this regard by getting the form generation to respond to CFD analysis results. There are also other researchers who already try to combine it with other environmental parameters, for example, energy with CFD simulation (Zhai and Chen, 2003), solar with wind (Capeluto et al., 2003) as well as various physical parameters (Murakami, 2006).
2. Related works

CFD simulations have been done for both outdoor and indoor environments. In both cases, the aim is to understand airflow so that heat gain can be minimised and to generate good cross ventilation. In the indoor environment, CFD is used for analysis to design for efficient ventilation system management (Somarathne et al., 2005) and ventilation systems for damage control (Steeman et al., 2009). In the outdoor environment, CFD has been used to look at the effects of heat gain from material, air-conditioning condensers and building geometry in the urban environment (Priyadarsini et al., 2008; Priyadarsini and Wong, 2005). We intend to look into the effects of CFD in the indoor environment as well as the effects of materials and air conditioning condensers when we proceed further into the research at a later stage.

Research has validated that CFD results are quite reliable by comparing with wind tunnel and site measurement results, although there are some differences under strong and mild wind conditions (Yoshie et al., 2007). Tominaga et al. (2008) did the same validations and found CFD techniques particularly useful for predicting wind environments in pedestrian areas.

With regard to our research, which focuses on high rise and high density environments, there are many strategies to plan and design well for good ventilation and airflow. These include façade design and articulations, window vent size, shape and location, envelope insulation, balcony provision, natural ventilation, centralised heating and cooling systems, building material selection and micro-environment design (Niu, 2004). In addition, building height response to heat (Takahashi, 2004), good city planning incorporating air paths (Ng, 2008a), smart building orientation in regard to prevailing wind (Ng, 2008b) will help to improve ventilation.

Still, most of the CFD results are done in two-dimensional mode and lack visualisation of how the air flow pattern looks like in 3D. We aim at doing three-dimensional CFD simulations to give a more complete picture since air flows are volumetric and three-dimensional. Most CFD research is also very independent on other inter-related environmental issues which we intend to address at a later stage of our research.

3. The site

Our site of interest for this initial CFD testing is a planning area situated at Kallang River, as shown in figure 2. It is one of the four growth areas identified in the Singapore Master Plan 2008. It is an inner urban area with a site area of 64 hectares of prime developable land, with the Kallang River running through it. The site is divided into four parts for which different themes have
been identified for their respective developments. The themes are: “Life in The Open,” “Life’s Beach,” “Buzz on the Green” and “A Unique Work–Live Place” (Urban Redevelopment Authority, 2009). We note that all the four themes focus on good outdoor living and recreation. In general, the planning is intended to create local environments that are conducive to outdoor activities and pedestrian linkages. Hence, it is crucial to have good ventilation in and around the site to support these themes. Given Singapore’s hot and humid tropical climate, it would be difficult to realise the planning vision to encourage people to want to come out and use the open areas if the air flow is poor.

The Urban Redevelopment Authority (URA) has already developed a simulated block model for the development area. The proposed plot ratio (floor space ratio) for the site as stated in the master plan ranges from 1.5 to 5.6. By referring to the block model, as well as taking the proposal building footprint and proposed plot ratio, we deduced the height of the buildings, which range from 3 to 36 stories.

4. The CFD process, result and validation

Figure 3 shows the entire CFD process from modeling to virtual reality (VR) visualisation. The CFD simulation requires the usage of solid modelling. There are several types of computer-aided design (CAD) software that do solid modeling, including AutoCAD and Revit Architecture. For the CFD simulation itself, there are a few well-known ones in the market, namely COMSOL Multiphysics as well as ANSYS Fluent and CFX. Since our research has just started, we experiment with COMSOL because it has the educational version, accepts major CAAD formats and far more affordable than ANSYS products. We managed to obtain CEI EnSight, which is the software for post-processing CFD virtual reality visualisation (Monroe et al., 2006) but it was too late to integrate this into the test for this paper.

Solid models have to be meshed (Asfour and Gadi, 2007) and scaled (Srebric et al., 2008) correctly for the simulation, as shown in figure 4. For the initial testing, we use the incompressible Navier–Stokes steady-state analysis
in 3D, which deals with incompressible isothermal fluid flow. We took four key climatic data for analysis, which are the annual mean temperature, the annual mean wind speed and direction, the air density and its dynamic viscosity of the annual mean temperature. The National Environment Agency provides the average temperature for the area, which is 26.925 degree Celsius, about 300 Kelvin. The air density is 1.184 kg/m$^3$ and dynamic viscosity is $1.983 \times 10^{-5}$ kg/ms at 300 Kelvin. There are two predominant wind directions from the north-east at 2.4 m/s and south-east at 1.9 m/s. The middle picture shows the site model being contained in a solid cuboid with the correct surface defined to represent the wind direction. The entire scene has to be meshed before CFD solving takes place.

The results of the CFD analysis are shown in figure 5. We can immediately notice that the current urban design highlighted with the square dotted line is creating a space where heat and air flow will be trapped, regardless of whether it is the northeast or southeast wind. We can thus respond with recommendations to alter the initial proposal based on these results so as to improve the ventilation through the site, especially at all the identified problem areas. Various planning and design variations can be generated and further tested.

After that, we did a validation in the HPC supercomputing environment to generate results using ANSYS Fluent as shown in figure 6. It validates the lack of air movement at the same zone. Although it can do much higher resolution airflow analysis in a shorter time, there is much work in constructing the 3D model in ANSYS Gambit (3D modeling tool) to export to Fluent as it does not support most CAAD 3D CAD formats. However, the benefit is more other smaller zones around buildings with stagnant airflow can be identified too because of the higher iterations that can be done in a shorter time.
Figure 5. Northeast (above) and southeast (below) wind flow analysis results.

5. Conclusion and future works

Each of the wind direction simulations in COMSOL took about an hour to be completed. This quick process of getting the wind flow is very encouraging and points to the great potential of integrating the process as an integral part of the urban design process. We can already understand the airflow behaviour and improve it at the early stage. Although we can get more accurate results by running more iterations, it is good enough for the conceptual stage. This will be a great improvement from the current process whereby ventilation issues are addressed only at the later architectural design stage when the planning guidelines are already determined.

Figure 6. Path lines coloured by velocity magnitude (m/s) of the northeast (left) and south-east (right) wind flow analysis in ANSYS Fluent 6.3. The shorter the lines are, the slower the airflow will be which means the zones are lacking natural ventilation.
Overall, it is getting more affordable, easier and quicker to generate useful results in the local desktops and workstations. Very soon, we will be able to do network CFD simulations in a practice or research work environment just as we do network rendering by harnessing the necessary computing and graphical power.

There are still other factors to test. Firstly, the ANSYS CFD and COMSOL to EnSight connection have to be explored thoroughly. ANSYS has now a framework called Workbench and modelling tool called DesignModeler to support more CAAD 3D CAD formats as well as running on Windows platform too but the license is still too costly. The potential of using VR software for CFD results for a more comprehensive overall visualisation will be explored, especially architects and urban designers are more inclined to view in 3D. They can analyse these results in 3D, propose alternatives and make various decisions before the actual design and implementation processes take place. Then, we need to explore the potentials of Ecotect Analysis’ airflow capabilities integration with the National Institute of Standards and Technology’s (NIST) Fire Dynamics Simulator and Smokeview (FDS-SMV) as well as EnergyPlus’ natural ventilation simulation. Another potential new CFD software to explore is Next Limit Technologies’ XFlow which may be faster to implement by using a particle-based, fully Lagrangian approach.

Apart from just measuring wind against urban structures, we would need to take into account more realistic settings where the presence of vehicles and trees canopy cannot be ignored (Mochida and Lun, 2008). We would also need to take into account pollutants, especially from transport (Tsou et al., 2001b), momentum and heat concentrations (Baik et al., 2003). Also, we must have to do the fluid with thermal incompressible flow interaction for transient analysis. With these, we would then be able to get a better overall picture of the airflow with thermal pattern for a sustained period of time (entire day, week, month or year) for the study area. The building facade materials will be taken into account for more realistic CFD simulation (Priyadarsini and Wong, 2005). Finally, the relationship between outdoor and indoor CFD analysis which is lacking in research will be explored too.

Harnessing the combined integration of graphics processing unit (GPU) and CPU power for parallel computing, which is supported by CFD is now a reality. This is the reason that we have equipped our hardware systems with NVIDIA graphics cards. The next step of our research is to connect all the bridges from modeling to VR visualisation of the CFD simulation results by using the CPU and GPU combination processing. NVIDIA Tesla and ATI Stream technology will give more mainstream users the capabilities to produce CFD simulations. The NVIDIA (Compute Unified Device Archi-
CUDA, which is a general purpose parallel computing architecture that leverages the parallel compute engine in NVIDIA GPUs to solve many complex computational problems in a fraction of the time required on a CPU, is another potential language to embark in to get the fullest potential possible out of the GPUs.

Apart from these, we should not forget about Intel’s revelation of Larrabee in the first quarter of 2010. Larrabee, which is a general-purpose computing on graphics processing unit (GPGPU), will be the third competitor in the high performance computing market. Indeed, this is an exciting time where CFD simulations will no longer be in the realm of the inaccessible supercomputers. Finally, testing the simulations in Windows 7 will also be another opportunity to be explored soon. We are keen to experiment with its multi-touch capabilities for VR visualisation.

Acknowledgements

The research project is funded by the Ministry of National Development, Singapore. We would like to thanks the lead specialist on High Performance Computing, Mr. Wang Junhong and chemical engineer Mr. Charles Hii Jun Khiong for their inputs on the CFD simulation.

References


