R. Stouffs, P. H. T. Janssen, S. Roudavski, B. Tunçer (eds.), *Open Systems: Proceedings of the 18th International Conference of the Association of Computer-Aided Architectural Design Research in Asia CAADRIA 2013,* 000–000. © 2013, The Association for Computer-Aided Architectural Design Research in Asia (CAADRIA), Hong Kong

COMPUTATIONAL FLUID DYNAMICS FOR ARCHITECTURAL DESIGN

Sawako KAIJIMA
Singapore University of Technology and Design, Singapore,
sawakokaijima@sutd.edu.sg

Roland BOUFFANAIS
Singapore University of Technology and Design, Singapore,
bouffanais@sutd.edu.sg

and

Karen WILLCOX
Massachusetts Institute of Technology, USA
kwillcox@mit.edu

Abstract. Computational fluid dynamics (CFD) is a cost-effective, well-known technique widely employed in industrial design. While indoor analysis can be achieved via CFD, wind tunnel testing (WTT) is still the prevailing mode of analysis for outdoor studies. WTT is often only performed a few times during the course of a building design/construction cycle and primarily for verification purposes. This paper presents a cross-disciplinary research initiative aiming to make CFD understandable and accessible to the architecture community. Our particular interest is in the incorporation of CFD into the early stages of architectural design. Many critical decisions, including those pertaining to building performance, are made during these stages, and we believe access to wind/airflow information during these stages will help architects make responsible design decisions. As a first step, we designed a passive cooling canopy for a bus stop based on the equatorial climatic conditions of Singapore where wind/airflow was a driving factor for geometry generation. We discuss our strategies for overcoming the two bottlenecks we identified when utilising CFD for this framework: mesh generation and result comprehension/visualisation.

Keywords. CFD; Simulation; visualization; concept design.

1. Introduction

Understanding natural phenomena in relation to buildings, particularly internal and external airflows, is becoming increasingly important to architectural design. This is due to the increased complexity of contemporary buildings and a growing interest in improving building performance in terms of the environmental impact.

2. Computational fluid dynamics in the building industry

Computational fluid dynamics (CFD) is a branch of fluid mechanics that utilises numerical methods to solve and analyse problems involving fluid flows. CFD has been commercially available since the early 1980s in the engineering community for applications such as turbo machinery, aerospace, combustion, and mechanical engineering. Today, CFD has proven to be a driving factor for performance enhancement in areas as diverse as Formula 1 racing, naval architecture for the America's Cup, and product development for swimwear; it has grown into an industry worth approximately 800 million dollars annually (Hanna, 2012).

During the past decade, CFD has been studied intensively in the domain of building environments. While the use of CFD for indoor applications is becoming established, further research is required for outdoor analysis (Blocken et al, 2009). Recent studies on the use of CFD for outdoor environments include simulation of pedestrian wind comfort (e.g. Mochida and Lun, 2008; Tominaga et al, 2008; Blocken et al, 2012), urban air pollution (e.g. Yang and Shao, 2008; Balczo et al, 2009; Tominiga and Stathopoulos, 2011), wind-driven rain (e.g. Huang and Li, 2010; van Hooff et al, 2011), and building surface heat transfer (e.g. Blocken et al, 2009; Defraeye et al, 2011; Karava et al, 2011). These studies and works focused on developing accurate and reliable simulation models and techniques or improving the design of relatively simple geometries with a few formal parameters.

Wind tunnel testing (WTT) remains the prevailing mode of analysis for outdoor studies in the building industry (Blocken et al, 2011). It is often performed only a few times during the course of a building design/construction cycle, if at all, for verification purposes. These tests are performed by wind/environmental consultants during the design development phase, and the impact of the gathered information on design is generally very limited.

The CFD versus WTT debate has been around since the introduction of CFD several decades ago; both methods provide a certain degree of knowledge and understanding of the environment in which the design exists. WTT requires an expensive setup and sophisticated instruments to measure field variables (wind velocity, pressure loads, turbulence intensity, and temperature). Its main limitation is that these measurements are obtained at only a few discrete points within the test section, which severely restricts understanding of the evolutionary or transient processes of unsteady complex phenomena such as vortex shedding, turbulence wakes, thermal stratification, and atmospheric boundary layer effects on the urban landscape.

CFD intrinsically overcomes this fundamental issue associated with WTT as the simulations yield instantaneous volume data. However, it suffers inherently from the discretisation of the governing equations of fluid dynamics – the Navier–Stokes equations for incompressible fluid flows – —combined with modelling of the initial and boundary conditions. Some flow phenomena exhibit an extreme sensitivity to these conditions, often referred to as the ‘butterfly effect’ – this effect can dominate flow dynamics, particularly at large Reynolds numbers where convective instabilities strongly affect the flow dynamics. These current limitations to using CFD are often misinterpreted as a major hurdle to its adoption as a standard practice in many industries. Even so, CFD is used successfully in the aerospace, automotive, and many product design industries (Figure 1); this fact alone stresses the compelling possibilities for the use of CFD in architectural design. However, little research has been done on the use of CFD in relation with the architectural design process, specifically in the early stages when many critical decisions, including those pertaining to the building performance are made (Bogenstätter, 2000; Bazjanac et al, 2011). We believe that access to the wind/airflow information during these early stages will assist architects in making responsible and fact-assisted design decisions.

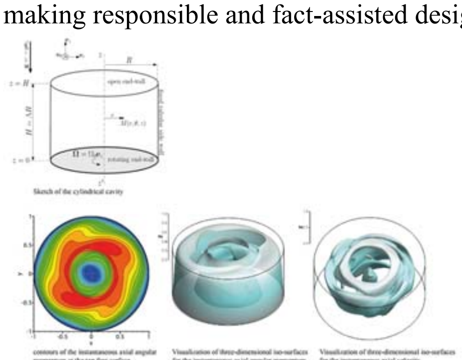

Figure 1. Use of CFD in engineering design. Simulation of an open transitional swirling (Bouffanais and Lo Jacono, 2009) flow.

3. ARCH-CFD: Case study context

In this context, ARCH-CFD was set up as a cross-disciplinary research initiative the International Design Centre established by the Singapore University of Technology and Design and the Massachusetts Institute of Technology (MIT), with the aim of making CFD understandable and accessible to the architecture community at large. The objective of this research initiative is not to extend knowledge in the field of CFD but to find ways of utilising CFD to support early stages of the architectural design process in synthesising the complex phenomenon of airflow along with its usually complex dynamics. As a first step, a passive cooling canopy for a bus stop was designed based on the equatorial climatic conditions of Singapore where wind/airflow was the main driving factor for geometry generation. The typical bus stop geometry found in Singapore was used as a benchmark for measuring the improvements of our design (Figure 2). Improvements were assessed visually and measured quantitatively based on probability density distribution of wind speeds and the Predicted Percentage Dissatisfied (PPD) (Zhai, 2006) and Thermal Sensation (TS) (Cheng, 2010). The NURBS modelling software Rhinoceros was used for architectural modelling, whereas ANSYS FLUENT was used for the CFD analysis part. Here, we discuss two bottlenecks identified when utilising CFD in this framework: mesh generation and result comprehension.

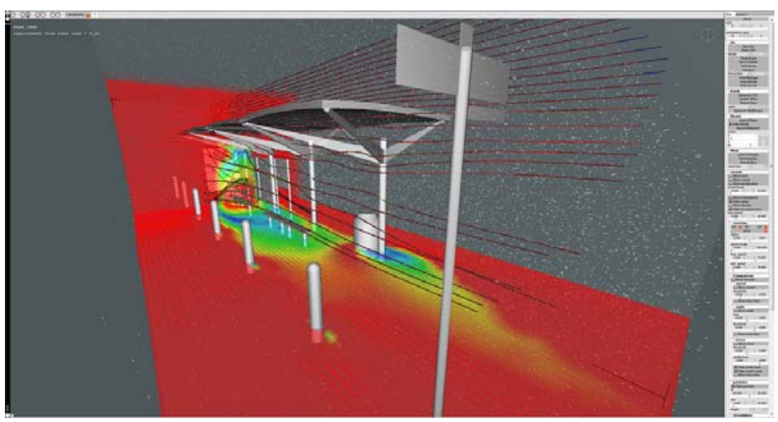

Figure 2. CFD analysis and flow visualization of an existing bus stop design using our in-house custom-developed visualization toolkit.

3.1. MESH GENERATION

Computer simulations involve modelling the reality of something as an abstraction to facilitate understanding towards a specific aspect of interest. As architects and engineers look into different aspects of reality, the models they develop and manipulate are not directly interchangeable. For instance, architects develop three-dimensional models with the aim of developing visual accuracy when communicating spatial qualities that often include mixtures of lines, surfaces, and solids. On the other hand, CFD practitioners' primary goal is to obtain numerically accurate analysis results, and common commercial tools require watertight solids as an input geometry.

Running CFD requires the creation of a volumetric meshing of the geometry of interest and its surroundings. This step is critical and is often the most labour-intensive. During the conceptual design phase, architects explore multiple geometries before arriving at a particular building design; this means that multiple meshing processes are required for repetitive runs of the CFD solver. Two aspects need to be balanced when meshing: quality and size. Mesh quality affects the overall accuracy of the analysis, while the mesh size – measured by the number of mesh nodes – dictates the overall computational cost, which can easily become overwhelming for complex geometries with extremely fine details.

At the current state of technology, architectural and engineering models are not interchangeable and require a significant amount of preprocessing for a simple standard conversion from one type to the other; a seamless conversion procedure is unfortunately still out of reach. Moreover, it is difficult to generalise a balancing strategy between simplification of geometry and accuracy of analysis. Typically, architects require quantity and speed rather than engineering accuracy from analysis in the early stages of design to explore multiple geometric options for compare and contrast. However, it is difficult for CFD practitioners to compromise on the accuracy as this could easily generate what they refer to as an 'untrusted' analysis.

In ARCH-CFD for our bus stop canopy case study, a hybrid mesh generation was employed to maintain an acceptable accuracy level while providing us with the flexibility of meshing various complex shapes and fine geometrical details (Figure 3). A hybrid mesh is the combination of a structured mesh (the outer surrounding environment) and an unstructured one (centred about the inner element of interest, i.e. the bus stop in our particular case of interest).

Historically, most computational methods were developed based on structured meshes, which, in general, are generated effortlessly. This is particularly true when boundaries have a simply geometry. However, with the advent of more powerful computers, CFD practitioners started simulating more complex flows and geometries for which structured meshes are commonly computationally prohibitive.

Unstructured meshes have gained popularity in recent years as an alternative approach for analysing flow dynamics involving complex geometries. Using an unstructured mesh offers two main advantages over traditional structured mesh. First, by not requiring the mesh to be logically rectangular, unstructured meshes offer considerable flexibility in discretising complex domains like the one we have, given the level of details of our bus stop. Second, and maybe of more importance in the framework of the ARCH-CFD project, unstructured meshes offer control of mesh density and stretching in a more flexible manner, which dramatically reduces the computational overheads. It is worth noting that those clear advantages come at a cost in terms of numerical accuracy. Thus our hybrid mesh strategy—using a coarse structured mesh in regions with limited geometrical complexity and an unstructured one near the bus stop—allows us to achieve a good trade-off between simulation accuracy and computational cost. More specifically, such a hybrid meshing strategy enables simple and rapid iteration of a particular conceptual design while maintaining a reduced level of mesh cells; this increases efficiency without increasing the computing cost.

Here, a parametric model for geometry generation was developed; this model omitted details such as small holes, fillets, and sharp corners that are small in relation to the overall domain size. While these details may be important from the standpoint of architectural expression, they have very little effect on the overall airflow. The parametric model ensured consistency of the model for data exchange from design to analysis and was used by the architects as a means to improve communication with the engineers regarding the range of geometries under consideration. However, towards the end of the design, we omitted the parametric model in order for architects to control the geometry without any parametric restrictions.

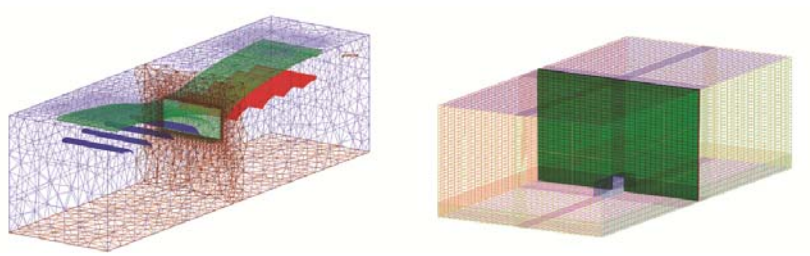

Figure 3. Hybrid mesh developed and used. Left: Inner unstructured mesh using tetrahedral cells around the geometry of interest (bus stop). Right: Outer structured mesh of the surrounding environment using quadrilateral cells

3.2. RESULT COMPREHENSION

Once a proper mesh has been generated and the CFD solver has completed its run, the next state, which is of paramount importance, is to be able to fully comprehend and appreciate the results obtained from the CFD analysis. Most architects are usually not familiar with CFD, so it is difficult for them to observe images provided by CFD practitioners and expand their understanding of wind/airflow.

To make CFD results more intuitive and amenable to architects, an interactive visualisation toolkit, originally developed by Kaijima and Michalatos in 2009, was adopted and further developed for this case study. The toolkit takes Rhinoceros models and ANSYS analysis results in an ASCII text format containing positions, wind velocity field (*x*-, *y*-, and *z*-components), and the turbulence kinetic energy field. The toolkit offers interactive 3-D visualisations of physical phenomena throughout the domain of interest by creating a voxel data structure from a hybrid mesh structure. The discrepancy between the ANSYS results and the custom toolkit visualisations is controlled by the voxel size to be unperceivable. In addition to the more typical streamlines or sectional visualisations, capabilities to view thermal comfort (Figure 4) and vorticity (i.e. the curl of the fluid velocity field connected to the local rotational rate of particles of fluids) are incorporated along with animated steady Lagrangian particle tracking to aid the user in appreciating and understanding the often counterintuitive airflow features throughout the entire domain (Figure 5).

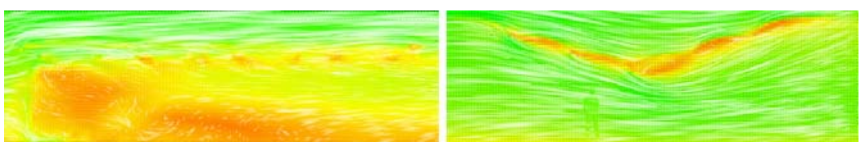

Figure 4. Sectional visualization of thermal sensation. Left: Existing bus stop design commonly encountered in Singapore. Right: Proposed bus stop design. The color map associated with the thermal discomfort goes from yellow to red; i.e. orange/red colors indicate areas of relative discomfort. TS= 0.1895TA-0.7754WS + 0:0028SR + 0:1953HR- 8.23 (Cheng, 2010) where TA = dry bulb air temperature, WS = wind speed, SR = solar radiation intensity, and HR absolute humidity

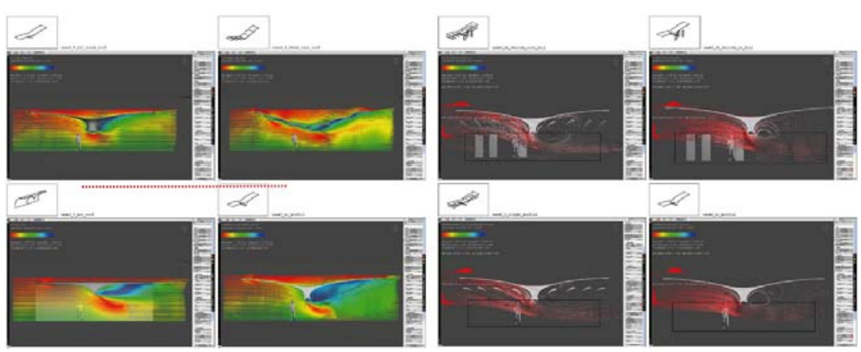

Figure 5. Examples of Design iterations. Screen Captures of the visualization toolkit.

The toolkit helped not only the architects but also the engineers in grasping the flow field in relation to the architecture geometry; overall, it greatly improved communication among the team, which ultimately resulted in an enhanced and improved design (Figures 6, 7). The details of quantitative as well as qualitative assessment method will be discussed in our future publications.

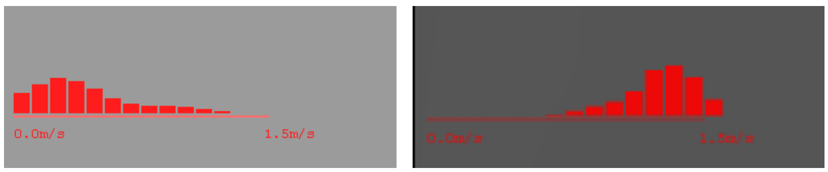

Figure 6. Probability Density Distribution of Wind speed. Left: Existing bus stop design commonly encountered in Singapore. Right: Proposed bus stop design.

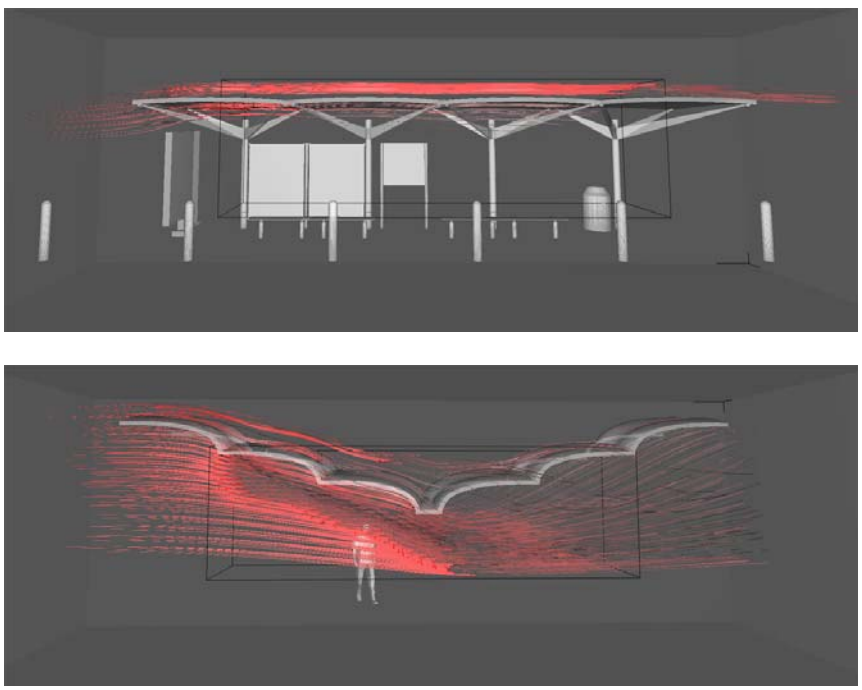

Figure 7 Visualization of a CFD analysis result using custom developed toolkit: Visualization of wind flow above 1.3 m/s coming into the space of interest. Top: Existing bus stop design commonly encountered in Singapore. Bottom: Proposed bus stop design.

4. Conclusions and outlook

Computer simulations such as CFD have opened up new possibilities for design and research by introducing environments in which we can manipulate and observe. However, using such simulation tools in a meaningful manner is not a straightforward or easy task. The aim of the bus stop canopy case study project was to build a platform that would facilitate domain knowledge exchanges within the existing framework as a first step.

While visualisation helped both parties – the architects and the engineers – in communicating and documenting the process, the meshing remained fairly time-consuming even for such a small case study project. Our strategy of using a hybrid mesh reduced the number of nodes to approximately 4% of the original unstructured mesh, yet the initial setup seems infeasible in a larger context. As the next step, we plan to use meshless CFD methods to improve the interoperability between the two domains and to simply eliminate one of the two bottlenecks mentioned above.

Acknowledgements

The authors would like to thank the financial support from the SUTD-MIT International Design Center for this interdisciplinary project.

References

Balczo. M.; Gromke, C., and Ruck, B.: 2009, Numerical modelling of flow and pollutant dispersion in street canyons with tree planting, *Meteorol., Z.* **18**(2), 197–206.

Bazjanac V.; Maile T.; Rose C.; O'Donnell J., and Mrazovic N.: 2011, An assessment of the use of Building Energy Performance Simulation in early design, *Proceedings of Building Simulation 2011*, Sydney, Australia, 1579-1585.

Blocken B.; Stathopoulos T.; Carmeliet J., and Hensen J.: 2009, Application of CFD in Building Performance Simulation for the outdoor environment, *Eleventh International IBPSA Conference*, Glasgow, Scotland, 489–496.

Blocken, B.; Stathopoulos T.; Carmeliet, J., and Hensen, J.L.M.: 2011, Application of CFD in building performance simulation for the outdoor environment: an overview. *J. Building Perform. Simul.,* **4**(2), 157–184.

Blocken, B.; Defraeye, T.; Derome, D., and Carmeliet, J.: 2009, High-resolution CFD simulations of forced convective heat transfer coefficients at the facade of a low-rise building. *Build. Environ*., **44**(12), 2396–2412.

Blocken B.; Janssen W.D., and Hooff T. van: 2012, CFD simulation for pedestrian wind comfort and wind safety in urban areasL General decision framework and case study for the Eindhoven University campus, *Environmental Modelling & Software*, **30**, 15–34

Bogenstätter, U.: 2000, Prediction and optimization of life-cycle costs in early design', *Building Research & Information*, **28**(9), 376–386.

Bouffanais, R. and Lo Jacono, D.: 2009, Unsteady transitional swirling flow in the presence of a moving free surface. *Phys. Fluids.,* 27, Art. 064107.

Chen Q. Y.: 2004, Using computational tools to factor wind into architectural environment design, *Energy and Buildings*, **36**, 1197–1209.

Cheng V.; Ng E.; Chan C., and Givoni B.: 2010, Outdoor thermal comfort study in subtropical climate: a longitudinal study based in Hong Kong, *Int. J Biometeorol*, **56**(1):43–56.

Defraeye, T.; Blocken, B., and Carmeliet, J.: 2011, Convective heat transfer coefficients for exterior building surfaces: Existing correlations and CFD modelling*, Energy Convers. Manage.,* **52**(1), 512–522.

Hanna K. R.: 2012, CFD in Sport – Retrospective 1 1992-2012, 9th Conference of the International Sports Engineering Association (ISEA), *Procedia Engineering*, **34**, 622–627.

Huang, S.H. and Li, Q.S.: 2010, Numerical simulations of wind-driven rain on building envelopes based on Eulerian multiphase model*, J. Wind Eng. Ind. Aerodyn.,* **98**(12), 843–857

Jakeman, A.J.; Letcher, R.A., and Norton, J.P.: 2006, Ten iterative steps in development and evaluation of environmental models, *Environmental Modelling & Software*, **21** (5), 602–614.

Karava, P.; Jubayer, C.M., and Savory, E.: 2011, Numerical modelling of forced convective heat transfer from the inclined windward roof of an isolated low-rise building with application to Photovoltaic/Thermal systems, *Appl. Therm. Eng*., **31**(11-12), 1950–1963.

Mochida, A. and Lun, I.Y.F.: 2008, Prediction of wind environment and thermal comfort at pedestrian level in urban area, *Journal of Wind Engineering and Industrial Aerodynamics*, **96** (10-11), 1498–1527.

Tominaga, Y.; Mochida, A.; Yoshie, R.; Kataoka, H.; Nozu, T.; Yoshikawa, M., and Shirasawa, T.: 2008, AIJ guidelines for practical applications of CFD to pedestrian wind environment around buildings, *J. Wind Eng. Ind. Aerodyn.,* **96**(10-11) 1749–1761.

Tominaga Y.; and Stathopoulos T.: 2011, CFD modelling of pollution dispersion in a street canyon: comparison between LES and RANS, *J. Wind Eng. Ind. Aerodyn*., **81**(1-3) 273–282.

van Hooff, T.; Blocken, B., and van Harten, M.: 2011, 3D CFD simulations of wind flow and wind-driven rain shelter in sports stadia: influence of stadium geometry, *Build. Environ*., **46**(1), 22–37.

Yang. Y. and Shao Y.: 2008, Numerical simulations of flow and pollution dispersion in urban atmospheric boundary layers, *Environmental Modelling & Software,* **23**(7) 906–921.

Zhai Z.: 2006, Application of Computational Fluid Dynamics in Building Design: Aspects and Trends, *Indoor and Built Environment*, **15**, 305–313.