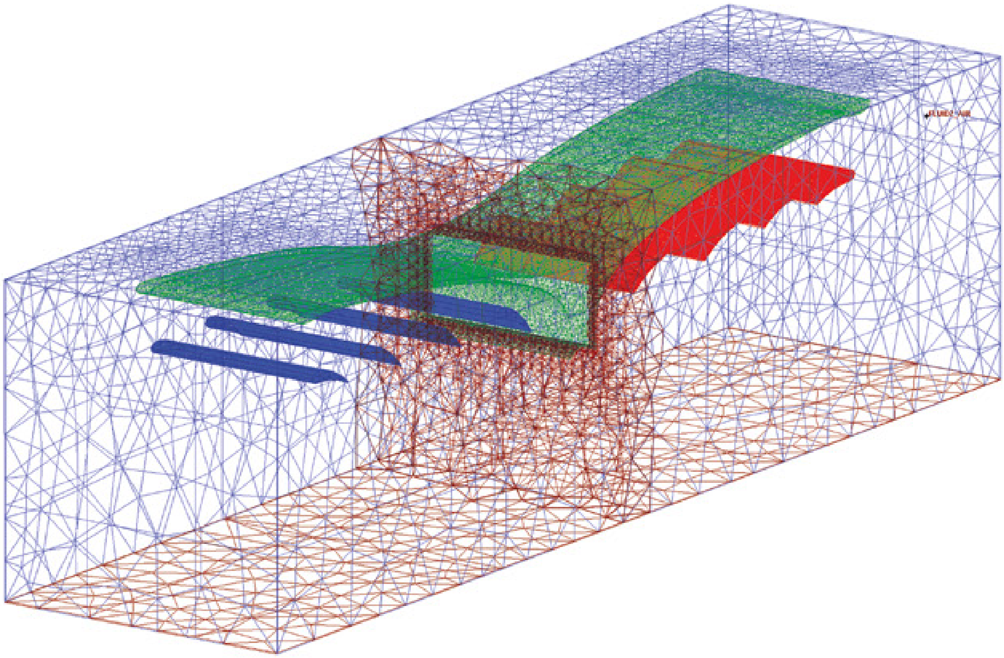
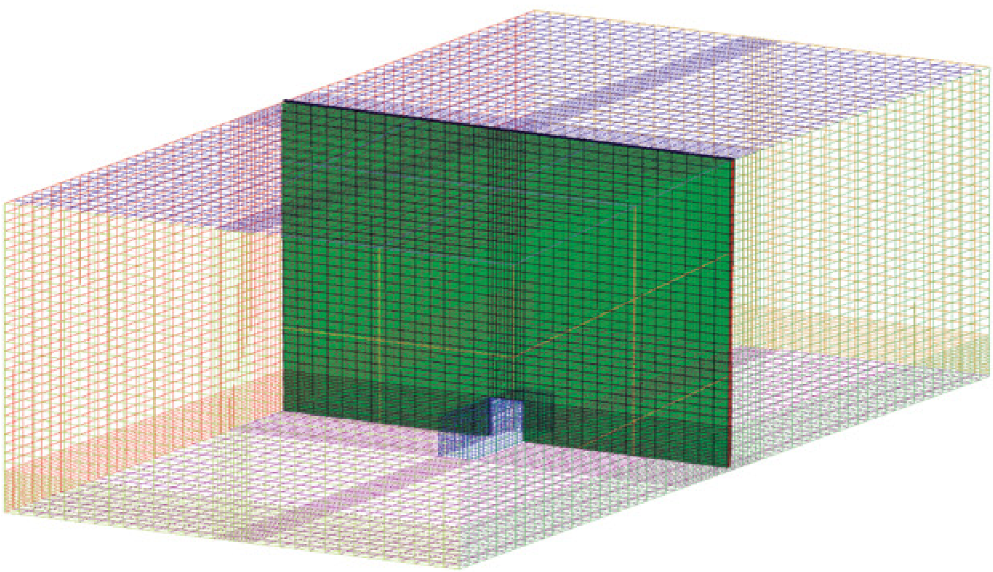


SAWAKO KAIJIMA, ROLAND BOUFFANAIS, KAREN WILLCOX AND SURESH NAIDU

There are many 'compelling possibilities' for computational fluid dynamics (CFD) in architecture, as demonstrated by its successful adoption in the aerospace, automotive and product manufacturing industries. **Sawako Kaijima, Roland Bouffanais, Karen Willcox and Suresh Naidu** of ARCH-CFD, a research initiative at the International Design Centre established by the Singapore University of Technology and Design (SUTD) and the Massachusetts Institute of Technology (MIT), explore CFD's potential.

COMPUTATIONAL FLUID DYNAMICS FOR ARCHITECTURAL DESIGN

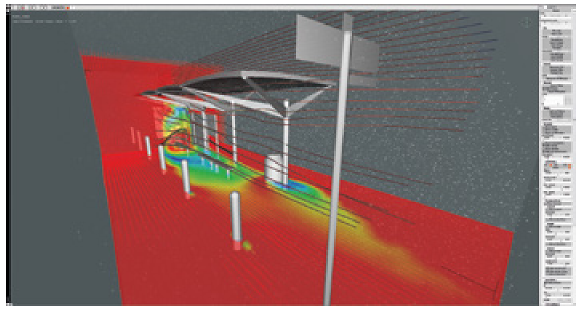
ARCH-CFD, Bus-stop canopy, International Design Centre, Singapore University of Technology and Design (SUTD) and Massachusetts Institute of Technology (MIT), 2012
Hybrid mesh. Opposite top: Unstructured mesh around the geometry of interest. Bottom: Structured mesh of the surrounding environment.



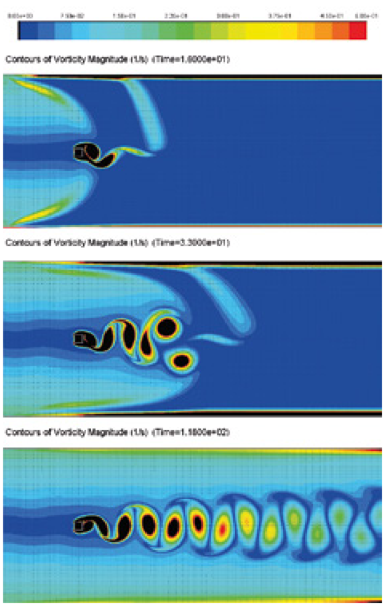
The understanding of natural phenomena in relation to buildings, and in particular internal and external air flow, is becoming increasingly important to architectural design. This is due to the increased complexity of contemporary buildings[1] and a growing interest in improving building performance in terms of environmental impact.[2]

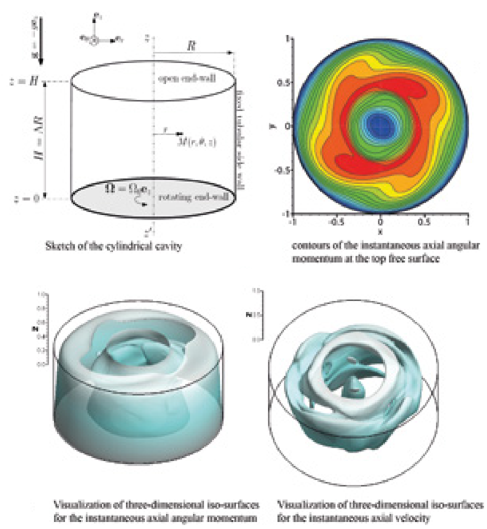
Computational fluid dynamics (CFD) is a cost-effective technique widely employed in industrial design. While indoor analysis can be achieved via CFD, for outdoor studies wind tunnel testing (WTT) is still the prevailing mode of analysis. Moreover, WTT is often performed only a few times during the course of a building design/construction cycle for verification purposes. 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, however, requires expensive setups 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, therefore severely restricting understanding of the evolutionary or transient processes of unsteady complex phenomena such as vortex shedding, turbulence wakes, thermal stratification, and the atmospheric boundary layer effects on urban landscape.



Computational fluid dynamics (CFD) is a cost-effective technique widely employed in industrial design. While indoor analysis can be achieved via CFD, for outdoor studies wind tunnel testing (WTT) is still the prevailing mode of analysis.





top: CFD analysis and result visualisation of an existing bus stop.

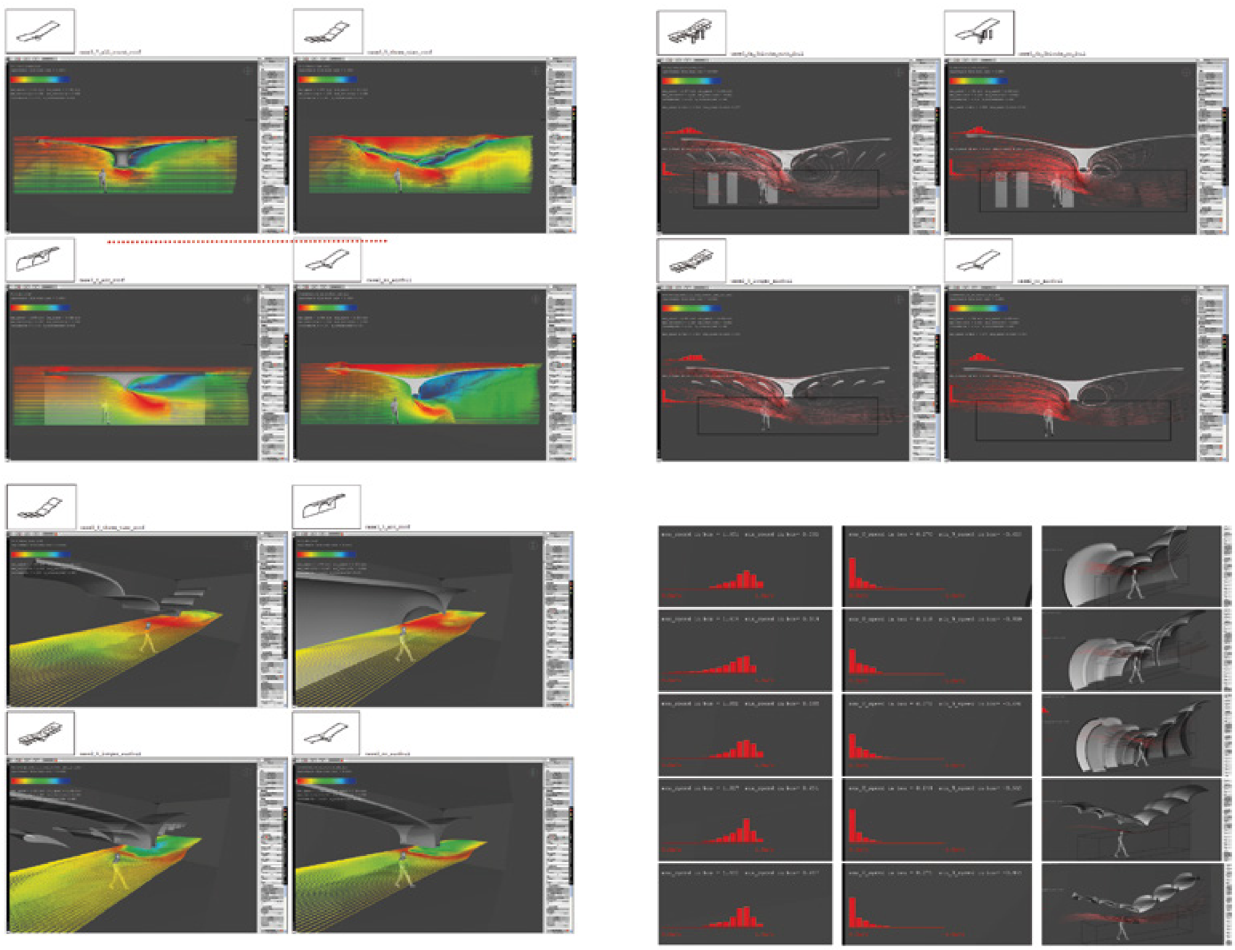
The use of CFD in engineering design
bottom right: Simulation of an open transitional swirling flow. Simulation of Shear-Driven Flows: Transition with a Free Surface and Confined Turbulence, Swiss Federal Institute of Technology, Lausanne, 2005–09.

bottom left: Vortex shedding. Simulation of the inversion of a von Kármán vortex street behind a confined cubical cylinder, Swiss Federal Institute of Technology, Lausanne, 2010–11.

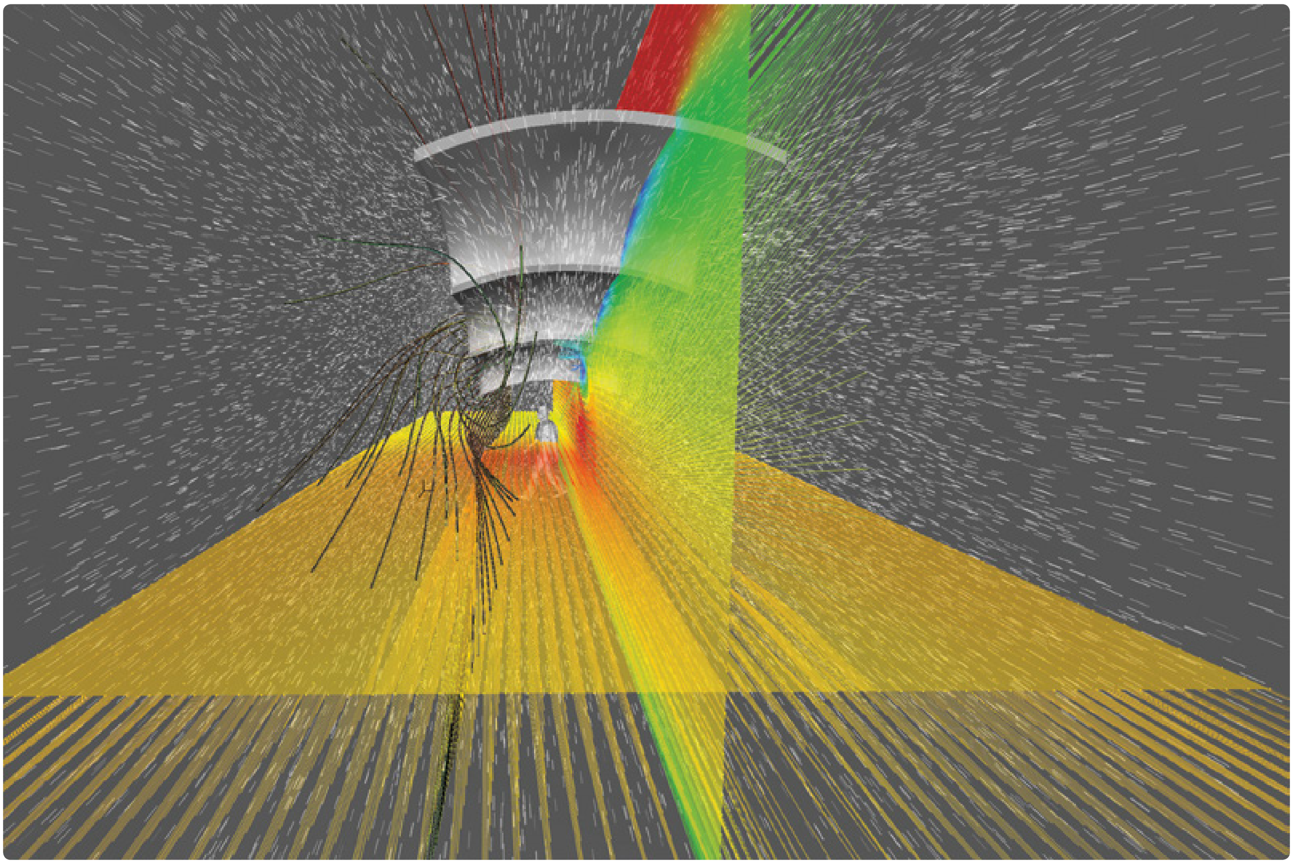
CFD intrinsically overcomes this issue as the simulations yield instantaneous volume data. However, it suffers inherently from the discretisation of the governing equations of fluid dynamics combined with the modelling of the initial and boundary conditions. Some flow phenomena exhibit an extreme sensitivity to these conditions, often referred to as the so-called ‘butterfly effect’.[3] These current limitations to using CFD are often misinterpreted as a major hurdle to its adoption as a standard practice in many industries. Yet CFD is used successfully in the aerospace, automotive and many product design industries; this fact alone stresses the compelling possibilities of CFD for architectural design.

THE ARCH–CFD PROJECT

ARCH-CFD is a cross-disciplinary research initiative at the International Design Centre established by the Singapore University of Technology and Design and the Massachusetts Institute of Technology (MIT), which aims to make CFD understandable and accessible to the architecture community. A particular interest is in the incorporation of CFD during the early stages of architectural design, when many of the critical decisions, including those pertaining to the general shape of buildings, are made. Access to wind/air-flow information during these early stages would help architects make responsible design decisions. As a first step in this research, a passive cooling bus-stop canopy has been designed based on a climatic condition of Singapore where wind/air-flow was a driving factor for geometry generation. Here, two bottlenecks were identified utilising CFD in this framework: mesh generation and result comprehension.



Design iterations. Screen captures of the visualisation toolkit.



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 simulations in a meaningful manner is not an easy task. The aim of the bus-stop canopy case study was to build a platform that would facilitate domain knowledge exchange within the existing framework as a first step of the ongoing research.

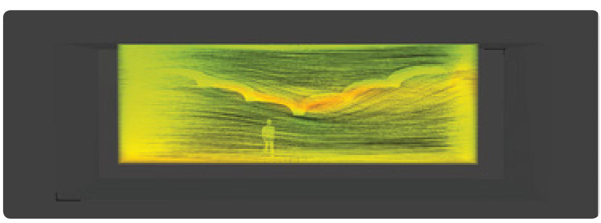
Visualisation of a CFD analysis result using the custom-developed toolkit.

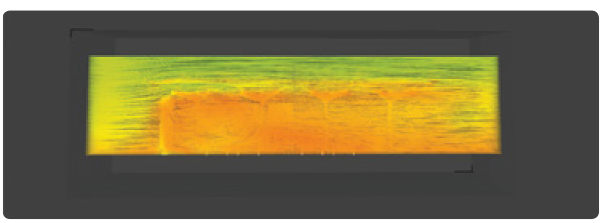
opposite: Visualisation of thermal comfort. Top: Proposed bus stop design. Bottom: Existing bus stop. Orange colour indicates areas of discomfort.

MESH GENERATION

Running CFD requires the creation of a volumetric meshing of the geometry of interest and its surroundings. This step is critical and the most manually intensive. During the conceptual design phase, architects explore multiple geometries before arriving at a particular building design, which means that multiple meshing processes are required to run CFD. There are two aspects that need to be balanced when meshing: quality and quantity. Mesh quality affects the overall accuracy of the analysis, while the quantity of mesh nodes dictates the computational cost, which can easily become overwhelming for complex geometries.

In the Arch-CFD bus-stop canopy case study, hybrid mesh generation is employed to maintain an acceptable accuracy level with the flexibility of meshing various complex shapes. Hybrid mesh is the combination of structured mesh (surrounding environment) and unstructured mesh (geometry of interest). This enables simple and rapid iteration of a particular conceptual design while maintaining a reduced level of mesh cells, therefore increasing efficiency while reducing the computing cost. Here, a parametric model for geometry generation was developed that omitted details such as holes, fillets and sharp corners that are small in relation to the overall size of the domain. While these details may be important for architecture expression, they have very little effect on the overall airflow. The parametric model ensured the 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.





RESULT COMPREHENSION

Subsequent to analysis is comprehension of the analysis results. Most architects are not familiar with CFD, and it is therefore difficult for them to observe images provided by CFD practitioners and to expand their understanding of wind/air-flow.

To make CFD results more intuitive for architects, an interactive visualisation toolkit, originally developed by Sawako Kaijima and Panagiotis Michalatos at Adams Kara Taylor, was adopted and further developed in the ARCH-CFD project. The toolkit takes analysis results in a text format containing position, wind speed, and turbulence kinetic energy and provides interactive 3-D visualisations of physical phenomena throughout the domain of interest. In addition to the more typical streamlines or sectional visualisations, capabilities to view thermal comfort and vorticity have been incorporated, as well as animated particle tracking to aid the user in understanding the often counterintuitive air-flow features throughout the domain. The toolkit helped not only the architects but also the engineers in grasping the flow field in relation to the architecture geometry, and overall greatly improved communication among the team, which ultimately resulted in an enhanced design.

Computer simulations such as CFD have opened up new possibilities for design and research by introducing environments in which we can manipulate and observe.[4] However, using such simulations in a meaningful manner is not an easy task. The aim of the bus-stop canopy case study was to build a platform that would facilitate domain knowledge exchange within the existing framework as a first step of the ongoing research. It is believed that the collective effort in this domain will soon make possible the use of CFD in the early stages of architecture, encouraging design decisions based on the underlying physics of air flow. ⌂

Notes

1. Christopher Alexander, *The Nature of Order: An Essay on the Art of Building and the Nature of the Universe*, Oxford University Press (New York), 1997.
2. Joana Carla Soares Gonçalves and Erica Mitie Umakoshi, *The Environmental Performance of Tall Buildings*, Earthscan (London), 2010.
3. Edward N Lorenz, 'Deterministic Nonperiodic Flow', *Journal of the Atmospheric Sciences*, Vol 20, No 2, 1963, pp 130–41.
4. Sherry Turkle, *Simulation and Its Discontents*, MIT Press (Cambridge, MA) 2009.