The use of CFD simulations for the assessment and improvement of a natural-ventilated hospital building

Alex Lee[a,*], Poh Hee Joo[a], George Xu Xiang Guo[a]; Kurichi Kumar[a], Andrew Meier[b]

[a]*Institute of High Performance Computing, Singapore;* [b]*CPG Consultants Pte Ltd, Singapore*

ABSTRACT
The objectives of the present work are to use computational fluid dynamics (CFD) to study the airflow pattern and to evaluate the effectiveness of the passive cooling design of the proposed Jurong General Hospital (JGH) using natural wind. The complex three-dimensional CFD model is used to assess the environmental conditions at the deep podium inside the JGH complex with prevailing wind conditions in Singapore, corresponding to the monsoon (November–February) and hot (March–October) seasons. The results include mean velocity vectors, streamlines and pressure distribution around the building as well as inside the podium space and walkway. It is shown that the effective depth for human comfort at the podium space may not be satisfied for all the prevailing wind conditions for the existing design. The effects of wind scoop, placement of wind baffle, openings, hybrid ventilation system, chamfering of sharp building edge (L corner) and many other novel architectural features are investigated.

INDEX TERMS
Natural ventilation; CFD; Indoor environmental quality

INTRODUCTION
With the increasing of energy cost; passive cooling of houses/commercial buildings using natural wind is coming back as an attractive alternative for mechanical air conditioning. Wind has been regarded as the most popular passive cooling resources against hot and humid climates in a tropical country, like Singapore. For buildings that are relatively large and complex, one further strategy is to only air-condition those working spaces, which necessitate full environmental control for comfort and to naturally ventilate transition spaces, such as corridors, lobbies and fire escape staircases, where possible.

In a naturally ventilated building, air is driven in and out due to pressure differences produced by wind or buoyancy forces. However, for large building with deep podium and complex building shape, achieving adequate human comfort to these naturally ventilated areas is a pressing issue. CFD has become one of the viable tools to assist both architect and mechanical engineer to design an efficient naturally ventilated building (Awbi, 1996; Gan, 2000; Varga and Oliveira, 2000; da Graça, 2002). In the design, the norms of earlier non-engineered tropical design (such as wind-paths and high ceilings or atria) are re-introduced and added to innovative modern features such as wind scoops or baffle to enhance IEQ (Indoor Environmental Quality).

METHODOLOGY
A commercial CFD software, FLUENT 6.0, was used in the CFD study. The numerical simulation was based on the conservation equation for continuity and momentum. As for the turbulence effect, a two-equation k–ε model was used. For solving a steady-flow problem, the four basic transport equations of the k–ε turbulence model in a 3D vector space are listed in Table 1.

[*] Corresponding author. E-mail: leekh@ihpc.a-star.edu.sg

Table1 Transport equations of the k–ε turbulence model in a 3D vector space

Transport equation	Mathematical expression in tensor format
Continuity	$\frac{\partial \overline{U_i}}{\partial x_i} = 0$
Momentum, $\overline{U_i}$	$\overline{U_i} \frac{\partial \overline{U_j}}{\partial X_i} = -\frac{1}{\rho} \frac{\partial \Pi}{\partial X_i} + \frac{\partial}{\partial X_j} \left[(\nu + \nu_t) \left(\frac{\partial \overline{U_j}}{\partial X_i} + \frac{\partial \overline{U_i}}{\partial X_j} \right) \right]$
Turbulence kinetic energy, $\overline{k}$	$\overline{U_j} \frac{\partial \overline{k}}{\partial X_j} = \frac{\partial}{\partial X_j} \left[\left(\nu + \frac{\nu_t}{\sigma_k} \right) \frac{\partial \overline{k}}{\partial X_j} \right] + \nu_t \left(\frac{\partial \overline{U_j}}{\partial X_i} + \frac{\partial \overline{U_i}}{\partial X_j} \right) \frac{\partial \overline{U_i}}{\partial X_j} - \overline{\varepsilon}$
Dissipation rate of turbulence kinetic energy, $\overline{\varepsilon}$	$\overline{U_j} \frac{\partial \overline{\varepsilon}}{\partial X_j} = \frac{\partial}{\partial X_j} \left[\left(\nu + \frac{\nu_t}{\sigma_\varepsilon} \right) \frac{\partial \overline{\varepsilon}}{\partial X_j} \right] + \frac{\overline{\varepsilon}}{\overline{k}} \left[C_1 \nu_t \left(\frac{\partial \overline{U_j}}{\partial X_i} + \frac{\partial \overline{U_i}}{\partial X_j} \right) \frac{\partial \overline{U_i}}{\partial X_j} \right] - C_2 \frac{\overline{\varepsilon}^2}{\overline{k}}$

There is also an ongoing experimental wind tunnel study for the present work, in collaboration with British Maritime Technology Limited, in order to validate the simulation results. Smith (1952) and Cermak *et al.* (1984) had previously evaluated the use of a wind tunnel to study natural ventilation. Interested readers can refer to their works.

BUILDING GEOMETRY

The design of the proposed Jurong General Hospital coincided with the greater emphasis on energy efficient and environmental friendly methods for engineering buildings. Figure 1 shows the artist impression of the building.

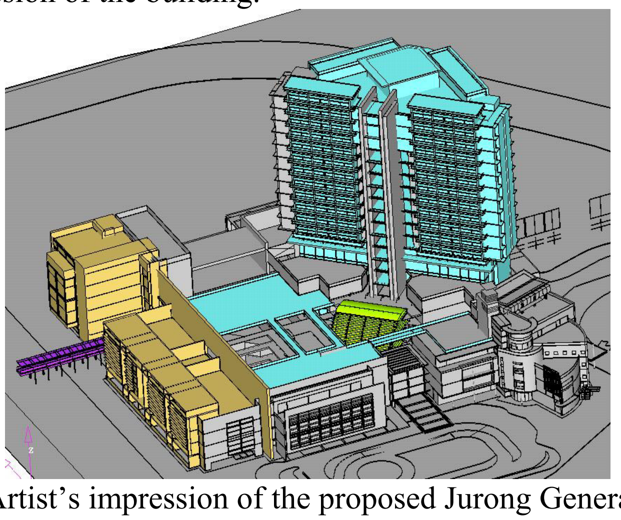

Figure 1 Artist's impression of the proposed Jurong General Hospital.

Figure 2 shows the simplified CFD geometrical model, with 2D-plan and 3D-isometric view, respectively. The site is measuring 140 m × 140 m on plan, with a four storey podium 32 m × 45 m on plan and a 15 storey tower block sitting atop the podium.

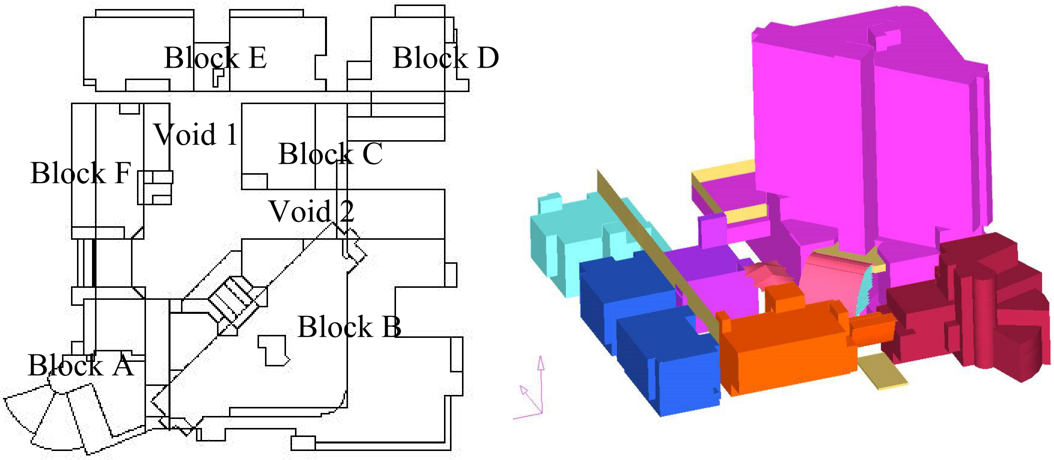

Figure 2 2D and 3D geometrical layout of proposed Jurong General Hospital.

RESULTS AND DISCUSSIONS

Two different wind scenarios were simulated, which are wind blowing from North to Northeast segment (predominant during the months of December to March) and wind blowing from South to Southeast segment (predominant during the months of June to August). Figure 3 shows the different scenario of wind blow direction.

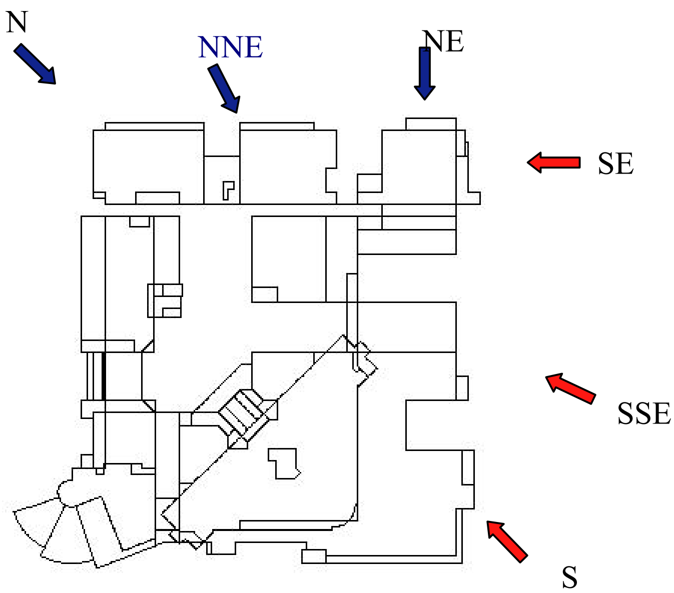

Figure 3 Different scenario of wind blow direction.

Results and Discussions for South to Southeast Scenario

Figure 4 shows the velocity contour plot at 1.5 m above the first storey level, as natural ventilation was critical in which public occupancy was highest in the common circulation routes and public foyers.

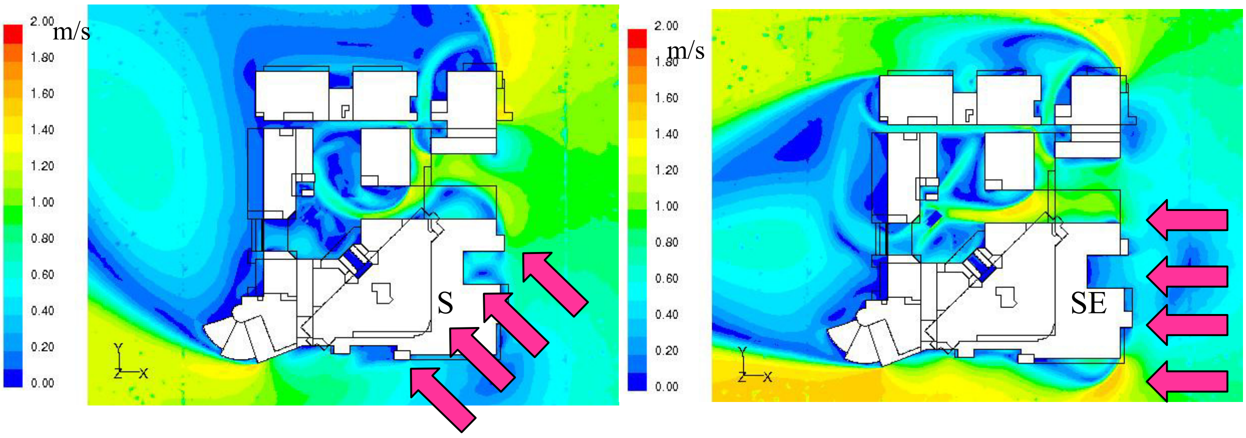

Figure 4 Velocity vector contour at 1.5 m above the first storey level for wind blow from South to Southeast scenario.

The large exit opening facing the SE direction is favourable for natural wind ventilation. However, the entrance opening is not aligned with the exit, and this creates substantial resistance to the airflow path and causes the ambient air losing its momentum to penetrate into the entrance. The deep podium is concealed from ambient wind flow path and huge air circulation is observed at that area. Therefore, it is suggested to taper the L-corner of block edge B and channel more ambient air to ventilate the podium area.

There is a slant scoop (angle of depression = 30°) situated between Blocks A and B. The purpose of having this passive, innovative feature is to depress the incoming ambient wind flowing downwards into the podium area. However, it is not able to ventilate the first level as the flow at level two is accelerated horizontally along the walkway at level two and misses out the podium at level one. This phenomenon is shown in Figure 5. One way to rectify this problem is to relocate the slant scoop further inside into the podium area in order to avoid the horizontal flow acceleration after the scoop, but rather to make the flow accelerated downwards into the podium.

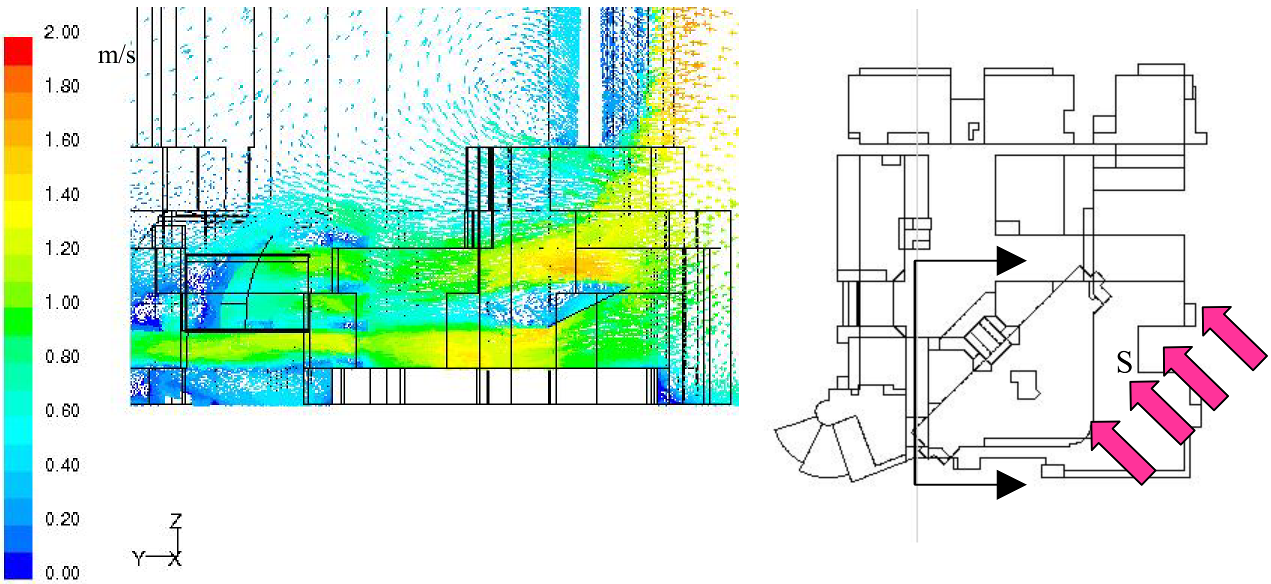

Figure 5 Horizontal acceleration of ambient wind along at level two after deflection by slant scoop.

Results and Discussions for North to Northeast Scenario

Again, the velocity contour plot at 1.5 m above first storey level is shown in Figure 6.

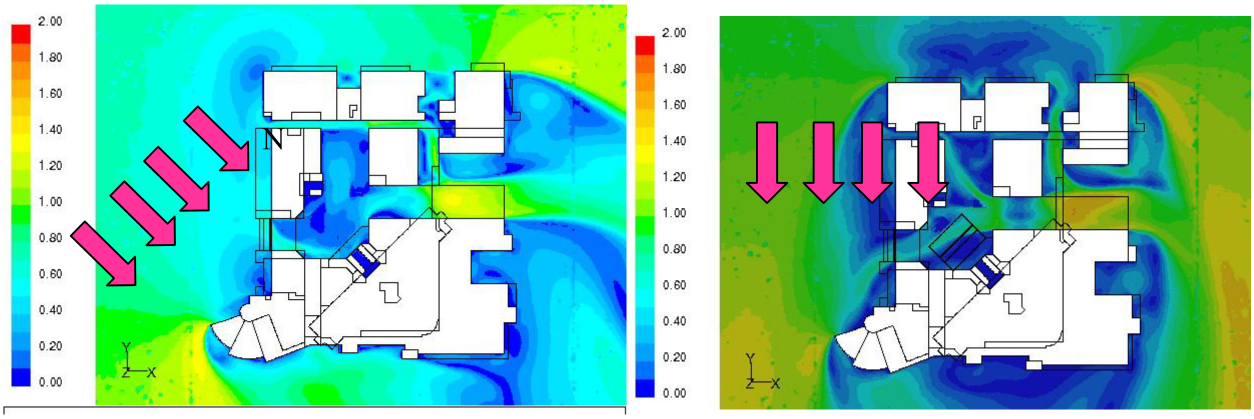

Figure 6 Velocity vector contour at 1.5 m above the first storey level for wind blow from North to Northeast scenario.

A closer investigation of the plot, especially after investigating the air flow trajectory format in 3D, would reveal that in void 2 area, ambient air is deflected downwards by the fifteen storey tower block (Block B) in front. After reaching the ground level, more air is channelled towards the Southeast direction, attributed to the larger opening and lower airflow resistance there. In view of the down-wash of ambient air through void 2, some architectural features can be made to channel more air flowing into the podium through void 2, such as cascading or staggering scoop or having slant facade between blocks B and C.

There is a wind corridor within Block E, which is covered at level one and having a slab on each subsequent level upwards. This design is to allow wind flowing into the podium, which is located about 50 m in front. However, from the simulation results shown in Figure 7, the ambient air flowing through the 8.1 m space does not have enough momentum to penetrate into the podium. Again, some slant scoops with angle of inclination about 30° to the horizon will be helpful to accelerate the ambient wind flowing into podium area, as demonstrated previously (Figure 5). Alternatively, one can re-align the wind passageway with the podium in order to minimize the flow restriction.

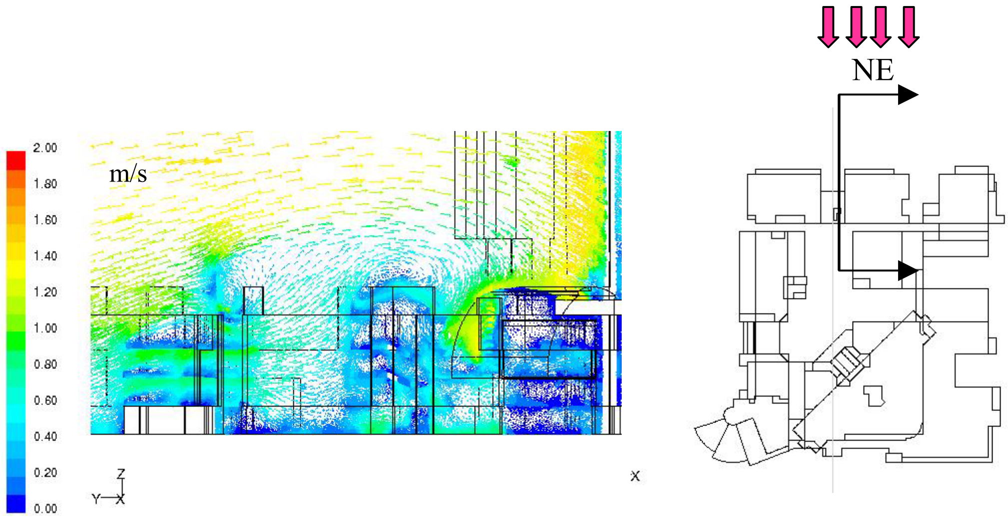

Figure 7 Velocity vector at cut plane across the podium during NE wind.

CONCLUSIONS AND IMPLICATIONS

The down-wash effects of the tower block on podium wind flow had been captured. Therefore, particular attention had to be paid for having a huge void below the high tower block, as architects can design some innovative feature such as slant facade to channel that huge amount of down-wash ambient air to ventilate the nearby podium. Besides that, the 'inlets' configuration relative to 'outlets' plays an important role to effectively ventilate the desired podium area. The architects have to re-align various building blocks properly in order to minimize flow restriction and avoid excessive wind shadow spots. Some localized plan configurations, such as chambering L-corner of building edge, will be useful to channel ambient air into appropriate area. This is due to the scientific phenomena of 'Coanda effect', which causes the flow to re-attach along the building surface. Having slant scoops at various 'inlet' locations would help to naturally ventilate the podium space, but one has to design it with appropriate angle of depression (e.g. 30°) and position with reference to podium space. In short, CFD had been demonstrated as a powerful tool to help architects and mechanical engineers to design a healthier building for natural ventilation.

ACKNOWLEDGEMENTS

The authors gratefully acknowledge the Healthcare Division, CPG Consultants Pte Ltd, Singapore, for their initiation and support to this work.

REFERENCES

Awbi, H.B. (1996). Air movement in naturally-ventilated buildings. *Renewable Energy* **8** (1–5), 241–247.

Cermak, J.E., Poreh, M., Peterka, J. and Ayad, S. (1984). Wind tunnel investigations of natural ventilation. *Journal of Transportation Engineering, ASCE Transactions* **110** (1), 67–79.

da Graça, G.C., Chen, Q., Glicksman, L.R. and Norford, L.K. (2002). Simulation of wind-driven ventilative cooling systems for an apartment building in Beijing and Shanghai. *Energy and Buildings* **34** (1), 1–11.

Gan, G. (2000). Effective depth of fresh air distribution in rooms with single-sided natural ventilation. *Energy and Buildings* **31** (1), 65–73.

Smith, E.G. (1952). The feasibility of using models for predetermining natural ventilation. Research Report #26, Texas Eng. Exp. Station, Texas A&M.

Varga, S. and Oliveira, A.C. (2000). Ventilation terminals for use with light pipes in buildings: a CFD study. *Applied Thermal Engineering* **20** (18), 1743–1752.

NOMENCLATURE

C_1, C_2, empirical constant in generation and destruction term of ε equation; 1.44 and 1.92, respectively

C_D, empirical constant for eddy viscosity, 1.0.

g_i, gravitational constant in X_i direction (m/s^2)

$\overline{k}$, mean turbulent kinetic energy (m^2/s^2)

$\overline{p}$, mean static pressure (N/m^2)

$\overline{U_{i,j}}$, mean velocity component in $X_{i,j}$ direction (m/s)

X_i, X_j, distance is Cartesian coordinate (m)

Greek Letter

$\bar{\varepsilon}$, mean dissipation rate of turbulence kinetic energy (m^2/s^3)

ν, kinematic molecular viscosity (m^2/s)

ν_t, eddy viscosity (m^2/s)

Π, total pressure (N/m^2); $\Pi = \bar{p} + \left(\left(2\rho\bar{k}\right) \big/ 3 \right)$

ρ, fluid density (kg/m^3)

σ_k, empirical constant of turbulent Prandtl number for k, 1.0

σ_ε, empirical constant of turbulent Prandtl number for ε, 1.22