THERMAL ENVIRONMENT AND AIRFLOW IN A SEMI-ENCLOSED SPACE SURROUNDED BY BUILDINGS – A NUMERICAL STUDY

Lina Yang†, and Yuguo Li
Department of Mechanical Engineering, the University of Hong Kong, Pokfulam Road, Hong Kong SAR, China

ABSTRACT
This paper presents a numerical study of the coupled airflow and thermal environment in a semi-enclosed space surrounded by buildings. Our numerical simulation couples the heat transfer calculation and the computational fluid dynamic (CFD) airflow simulation in which a RNG (renormalization-group) $k-\varepsilon$ model is used. Both the solar radiation and building/ground thermal storage were considered in the heat transfer part. The heat transfer model needs the convective heat transfer information given by CFD and at the same time it can also provide surface temperature as the thermal boundary condition for CFD simulations. Through this model, the heat transfer and airflow behaviors in the buildings and the surrounding environment can be investigated. In addition, the effects of different building/ground materials were analyzed through the coupled airflow and thermal model. It is found that the building/ground thermal mass and its thermal properties can have significant influences on the thermal environment around the buildings. The results can provide some guidance on improving thermal environment, airflow information, and as well as the urban design and building plan for architects and building engineers.

KEYWORDS
Thermal environment, Airflow, Heat transfer, CFD, RNG $k-\varepsilon$ model

INTRODUCTION
With the rapid urbanization in developing countries, many urban cities are becoming denser, higher and more compact with a limited sky view factor. The heat and pollutant exhaust from buildings and vehicles can be more easily accumulated in the limited surrounding semi-enclosed space. The surrounding air environment around buildings, closely related to indoor environment, can also affect public health, the environment, and the amount of energy used for heating or cooling in buildings. The study on indoor environment may no longer be studied in isolation. Air movement/natural ventilation induced by wind or thermal buoyancy has been used as a passive cooling strategy for improving the thermal comfort or urban environment (Capeluto 2005). Various numerical simulations, wind tunnel tests and laboratory/field experiments have been conducted to investigate the ventilation and its ability for pollution dispersions in urban street canyons (He et al. 1997, Chang and Meroney 2003, Georgakis and Santamouris 2006). However, in dense cities, buildings in urban areas are usually high and the open space or void is relatively small and compact, which are considered to be important barriers to the wind-induced ventilation potential. Under calm conditions, the air movement by thermal buoyancy becomes more important for urban environment. Heat transfer and airflow behavior studies have become crucial parts for urban design and planning to create comfortable urban environment. However, research into integrated airflow and thermal environment in urban cities is limited. These studies need to combine thermal mass, solar radiation, heat transfer between buildings/ground and ambient air, and the detailed airflow movement. The lacks may become the potential barriers to the good

† Corresponding Author: Tel: + 852 96076926, Fax: + 852 28585414
E-mail address: lnyang@hkusua.hku.hk
urban planning and design. To fill up the gap, both the heat transfer model and the CFD code were coupled to simulate the thermal environment and airflow around buildings in urban region. The integrated model has been found to be a useful evaluation and design tool for architects and engineers.

MODEL DESCRIPTION
For simplicity, a simple urban model - a semi-enclosed space surrounded by buildings is considered here. Figure 1 showed the model geometry and computational domain. Such a semi-enclosed space enclosed by buildings, as one form of urban pattern, can be a courtyard, a square, a park or a lake surrounded by buildings etc. The dimension of the semi-enclosed space considered here is $3H \times 3H \times H$ with the surrounding building of $2H$ depth (Height: width = 1: 3 for the semi-enclosed space). The whole domain is $22H \times 17H \times 4H$.

For the model, almost all the heat transfer processes can be involved. The ambient air enters in and internal air goes out from the semi-enclosed space through the open “roof” by natural forces (stack effect and/or wind induced). The thermal mass of the buildings and the semi-enclosed space floor can absorb or release the heat through its body. The heat can penetrate through the thermal mass body by heat conduction and the heat convection can occur between the surface and surrounding air. In addition, the solar radiation can reach into the model at different positions and different time. The radiative heat transfer between model surfaces cannot be neglected as its different temperature with different solar radiation.

To obtain a good understanding of thermal environment and airflow behavior, two parts should be included: one is the heat transfer model for predicting the thermal behavior; and another is computational fluid dynamics model for calculating the air flow. Both of them are very important tools and should be coupled together to predict the thermal environment and airflow around buildings.

HEAT TRANSFER MODEL (HT)
All the surfaces of the model were divided into small surface elements for heat transfer calculation. For each element, solar radiation, long-wave radiation between building/ground surfaces, sky radiation, convective heat transfer between building/ground and ambient air nearby, and heat conduction through it should be taken into account.

Shape factor calculation
The shape factor of two surfaces (i.e. configuration factor or view factor) is defined as the fraction of
energy leaving one surface element that arrives at another one (Siegel and Howell 1972). The discrete transfer method (DTM), which can trace the rays from one surface to another, is used for its applicable to complex geometry. The DTM is based on the concept of solving the radiation transfer equation for representative rays and to this extent it is related to the Monte Carlo model. But the directions of the rays are specified and not chosen at random. So the model is numerically exact, geometrically flexible and easily coupled to a CFD solver (Cumber 1995). These attractive virtues have made the discrete transfer method popular with many heat transfer applications (Cumber 1995). The detail description of DTM can be found in Lockwood and Shah (1981).

**Gebhart’s absorption factors calculation**

Gebhart’s absorption factors provide the fraction of energy emitted by a surface that is absorbed at another surface after reaching the absorbing surface by all possible paths (Gebhart 1971). For long-wave radiation and short-wave radiation, the values of Gebhart’s absorption factors are different due to different absorptivity. They were calculated separately for each surface element.

**Solar radiation calculation**

As mentioned before, the DTM method can be used to calculate shape factor for any complex geometry. With good capability to see if two surface can be viewed each other, it can also be used to judge if any given surface can be shined by the sun or not. For each surface element, the direction of ray is specified as opposite to the sun incidence direction. Along the path, if the ray impinges on other blocks or objects, it means the sun’s rays cannot reach to this surface and the surface is sunshade; otherwise, it is sunlit. The direct irradiance can be calculated by the Bouguer’s law and the diffuse irradiance can be based on the Berlage’s formula (Yan and Zhao 1986).

**Sky model**

Berdahl and Martin’s sky temperature model (Duffie and Beckman 1991) is adopted here. This model relates sky temperature $T_s$ (K) to the dry bulb temperature $T_a$ (K), dew point temperature $T_{dp}$ (°C), and hour from midnight $t$ (h).

**Heat conduction model**

For simplicity, we assume the ground or wall to be homogeneous with a constant thermal diffusivity for the same media separately. The ground model considered here is a slab-on-grade pattern. The one dimensional transient heat conduction equation is:

$$\frac{\partial T}{\partial t} = \alpha \frac{\partial^2 T}{\partial x^2}$$

where $\alpha$ is the thermal diffusivity.

The boundary conditions are given according to the heat balance on both external and internal surfaces. For the external wall or ground adjacent to the semi-enclosed space air:

$$K \frac{\partial T}{\partial x} = h_{cr}(T_s - T_a) - \sum q_e$$

where $\sum q_e = q_{s}^+ + q_{s}^- + q_{L}^+$ is the net radiation heat gain, including the direct solar radiation and diffuse solar radiation heat gain $q_s^+$, long-wave radiation from other building/ground surfaces and sky radiation heat gain $q_L^+$, and radiation heat loss $q_L^-$ by itself. $K$ is the heat conductivity of wall or ground, $h_{cr}$ is the external convective heat transfer coefficient (CHTC) and can be given by CFD. This value is a very important parameter for estimating accurate convective heat transfer from the surface.

For the internal wall adjacent to the indoor air:

$$K \frac{\partial T}{\partial x} = -h_i(T_s - T_{in})$$
where $T_0$ is the indoor air temperature and assumed to be constant due to the air conditioning system. $h_{gb}$ is the internal CHTC and can be given by empirical value.

For the bottom of the ground, the temperature is assumed to be constant, i.e. the average annual soil temperature $T_0$.

The calculation procedures of HT program can be shown as Figure 2a. Firstly, specify the initial conditions including the grid information, and the basic thermal properties; secondly, for each element, calculate the shape factor by using DTM, calculate the Gebhart’s absorption factor for long-wave and short-wave radiation separately; thirdly, evaluate the sunlit or sunshade by DTM and calculate the solar radiation for each element; and finally, according to the heat balance in the boundary, calculate the surface temperatures through heat conduction, provided that the CHTCs and ambient air temperatures near surface elements are given by CFD.

AIRFLOW MODELLING

The RNG $k-\varepsilon$ turbulence model proposed by Yakhot and Orszag (1986) was chosen to consider the effect of turbulence. The equations for three-dimensional, steady and turbulent flow include:

Continuity equation

$$\frac{\partial (\rho u)}{\partial x} + \frac{\partial (\rho v)}{\partial y} + \frac{\partial (\rho w)}{\partial z} = 0 \quad (4)$$

Momentum equations

$$\frac{\partial (\rho u)}{\partial x} + \frac{\partial (\rho u v)}{\partial y} + \frac{\partial (\rho u w)}{\partial z} = -\frac{\partial p}{\partial x} + \frac{\partial}{\partial y} \left( \mu + \rho \mu \right) \frac{\partial u}{\partial y} + \frac{\partial}{\partial z} \left( \mu + \rho \mu \right) \frac{\partial u}{\partial z} \quad (5)$$

$$\frac{\partial (\rho v u)}{\partial x} + \frac{\partial (\rho v v)}{\partial y} + \frac{\partial (\rho v w)}{\partial z} = -\frac{\partial p}{\partial y} + \frac{\partial}{\partial x} \left( \mu + \rho \mu \right) \frac{\partial v}{\partial x} + \frac{\partial}{\partial z} \left( \mu + \rho \mu \right) \frac{\partial v}{\partial z} \quad (6)$$

$$\frac{\partial (\rho w u)}{\partial x} + \frac{\partial (\rho w v)}{\partial y} + \frac{\partial (\rho w w)}{\partial z} = -\frac{\partial p}{\partial z} + \frac{\partial}{\partial x} \left( \mu + \rho \mu \right) \frac{\partial w}{\partial x} + \frac{\partial}{\partial y} \left( \mu + \rho \mu \right) \frac{\partial w}{\partial y} \quad (7)$$

Energy equation

$$\frac{\partial (\rho T)}{\partial x} + \frac{\partial (\rho u T)}{\partial y} + \frac{\partial (\rho w T)}{\partial z} = \frac{\partial}{\partial x} \left( \alpha + \frac{\partial T}{\partial x} \right) + \frac{\partial}{\partial y} \left( \alpha + \frac{\partial T}{\partial y} \right) + \frac{\partial}{\partial z} \left( \alpha + \frac{\partial T}{\partial z} \right) + \frac{Q_T}{c_p} \quad (8)$$

Turbulence kinetic energy

$$\frac{\partial (\rho k)}{\partial x} + \frac{\partial (\rho u k)}{\partial y} + \frac{\partial (\rho w k)}{\partial z} = \frac{\partial}{\partial x} \left( \alpha + \frac{\partial k}{\partial x} \right) + \frac{\partial}{\partial y} \left( \alpha + \frac{\partial k}{\partial y} \right) + \frac{\partial}{\partial z} \left( \alpha + \frac{\partial k}{\partial z} \right) + G_k + G_w \quad (9)$$

Dissipation rate of turbulence kinetic energy

$$\frac{\partial (\rho \varepsilon)}{\partial x} + \frac{\partial (\rho u \varepsilon)}{\partial y} + \frac{\partial (\rho w \varepsilon)}{\partial z} = \frac{\partial}{\partial x} \left( \alpha + \frac{\partial \varepsilon}{\partial x} \right) + \frac{\partial}{\partial y} \left( \alpha + \frac{\partial \varepsilon}{\partial y} \right) + \frac{\partial}{\partial z} \left( \alpha + \frac{\partial \varepsilon}{\partial z} \right) + C_1u \frac{e^2}{k} \left( \frac{G_k + C_w G_w}{\varepsilon} \right) - C_2u \frac{e^2}{k} \quad (10)$$

where $\mu_{eff} = \rho \mu_{eff}$, $C_w = 0.0845$, $C_a = 1.42$ and $C_p = 1.68$.

A finite volume method is used to discretize the governing equations on a non-staggered grid system (Patankar 1980). The HYBRID scheme is used for discretizing the convection terms in the momentum
equations, turbulent kinetic energy, dissipation rate and energy. The SIMPLEC algorithm is used for the coupling between velocity and pressure to obtain the pressure field in the iteration procedure (Patankar 1980). The convergence of the numerical solution is monitored by checking the residual of all the governing equations. The accuracy of CFD results is highly sensitive to the boundary conditions. The heat flux or surface temperature is used as boundary conditions for solving energy equations.

**COUPLING BETWEEN HEAT TRANSFER AND AIRFLOW**

The coupling approaches

The previous sections have shown that convective heat transfer is very important for both HT and CFD. On the one hand, CFD can provide CHTC and ambient air around buildings for HT; on the other hand, HT can use the information given by CFD to calculate the surface temperature for CFD as boundary conditions. Therefore, it is necessary to integrate HT and CFD together to obtain the thermal environment and airflow around buildings. The coupling between these two programs is to exchange the convective heat transfer information each other. As the mesh for calculating airflow in CFD code is different from that used in HT program, the grid match should also be considered in the coupling process. Usually the mesh size of HT is larger than CFD.

As described in the Figure 2b, the detail coupling procedures can be divided into four parts: 1) Given proper initial surface temperatures, calculate the airflow through CFD code; 2) the CHTCs and surrounding air temperatures near surface obtained by CFD are used to obtain the new surface temperatures through HT program; 3) re-calculate the airflow in CFD by applying the new temperature boundary conditions; and 4) check the convergence of airflow and energy, if not yet, repeat the step 2 and 3 until the convergence is achieved.

**CHTC calculation**

As mentioned above, it is very important to evaluate CHTC accurately. The standard wall function is used here for this purpose. It has shown that standard wall function is inadequate for modeling natural convection in rooms because the results are not independent of near-wall grid resolution (Schild 1997). An alternative approach is to use the empirical formulas. However, the information about external CHTC is still limited and unavailable. More studies and experiments on this problem are required in future for predicting good simulation results.

**TESTS OF PROGRAM**

The steady test has been conducted through giving certain parameters (see Table 1). Velocity inlet boundary was used for the inlet of domain with a logarithmic boundary layer in the neutral atmosphere by wind tunnel experiment (Soulhac 2000). The symmetry boundary (zero flux of all quantities) was used at the top surface and lateral surfaces. All the grounds and building walls were specified as no-slip condition. Standard wall functions were adopted.
Table 1. Initial conditions and thermal properties for model test

<table>
<thead>
<tr>
<th>Thermal properties</th>
<th>Slab</th>
<th>Soil</th>
<th>Wall</th>
</tr>
</thead>
<tbody>
<tr>
<td>Depth (D - m)</td>
<td>0.4</td>
<td>5.0</td>
<td>0.2</td>
</tr>
<tr>
<td>Thermal conductivity (K - W/m.K)</td>
<td>0.3</td>
<td>1.0</td>
<td>0.5</td>
</tr>
<tr>
<td>Volumetric heat capacity (ρc - MJ/m³)</td>
<td>1.0, 2.0, 4.0</td>
<td>5.5, 0.42, 0.84, 1.68</td>
<td></td>
</tr>
<tr>
<td>Radiation parameters</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Emissivity (ε)</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Reflectivity (ρ)</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Transmissivity (τ)</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Ground/building (Long-wave)</td>
<td>0.9</td>
<td>0.1</td>
<td>0.0</td>
</tr>
<tr>
<td>Ground/building (Short-wave)</td>
<td>0.55, 0.75, 0.9</td>
<td>0.45, 0.25, 0.1</td>
<td>0.0</td>
</tr>
</tbody>
</table>

Note: * is the standard case.

Figure 3 and 4 show the surface temperature contours and velocity vectors in the center x-z plane for different cases. For these cases, almost all horizontal surfaces obtain the same solar radiation. Higher surface temperatures are caused by relatively smaller convective heat transfer with ambient air in these regions. A clockwise rotating vortex flow as shown in the flow pattern figure was developed inside the semi-enclosed space. Inside the space, the solar radiation of leeward building walls is larger than windward sides, which can reinforce the vortex circulation due to the thermal buoyancy.

From the Figure 3, we compared the temperature distribution and airflow pattern with different shortwave emissivity. From the figure, we can find that the surface shortwave emissivity is an important parameter. A higher shortwave emissivity means higher shortwave absorption and lower reflection (i.e. albedo), which can absorb more solar radiation. Subsequently the surface temperature will increase due to more heat gain according to the surface energy balance. The temperature field in the center x-z plane showed that the higher surface temperature can also visibly influence the surrounding air temperature distribution and air movement due to the heat convection with adjacent air (see Figure 3).

(a) $\varepsilon = 0.55$
(b) $\varepsilon = 0.75$ (standard case)
(c) $\varepsilon = 0.9$

Figure 3. Temperature contours and velocity vectors for different surface short-wave emissivity

In Figure 4, different slabs and walls with different volumetric heat capacities are compared. It is obviously that thermal mass or heat capacity play an important role on the heat penetration through the walls/slabs. Since only steady turbulent flow is considered, the time delay effects of thermal mass could not be presented here. However, the effect of thermal storage or heat penetration capability can be found according to the simulations. Heavier building walls and ground covers can decrease the surface
temperature and surrounding air temperature slightly due to its more powerful heat absorption.

![Temperature contours and velocity vectors for different volumetric heat capacities (MJ/m³)](image)

Based on these limited simulated results above, we can find that the thermal properties of buildings or ground have significant effects on the surrounding thermal environment and air flow. Reducing the short-wave emissivity and increasing thermal capacity of buildings/ground may reduce the surface temperature and improve the thermal comfort nearby. Due to the space limit, the effects of other thermal properties of buildings/ground were not presented here. Therefore, it is necessary to choose proper materials and utilize the limited open space for improving our urban environment. Although the effects of moisture are not involved at present, this coupling model can still simulate the thermal environment and air movement by considering the major heat transfer behaviors. This preliminary study may give a hope to investigate in detail the effect of urban elements (buildings and covers) on the urban environment through the combination model of heat transfer and CFD. It can be a useful tool to provide information of space utilization, as well as a good guide for practical city planning and building design.

CONCLUSIONS

Through developing a numerical model with the coupling of heat transfer and CFD, a simple model—a semi-enclosed space surrounded by buildings has been considered to investigate the urban thermal environment and air movement. The coupled model has been shown preliminarily to be useful to analyze urban environment for proper urban planning and design. The results showed that different semi-enclosed space with different thermal mass may cause different thermal environment. To simulate the complex microclimate in urban areas, the effect of moisture should be considered in future. Further studies on CHTC and experiments are required.

ACKNOWLEDGEMENTS

The work described in this paper was supported by a grant from the Research Grants Council of the Hong Kong Special Administrative Region, China (Project No. HKU 7154/05E).

NOMENCLATURE

- \( c \) specific heat capacity (J/Kg)
- \( T_{dp} \) Dew point temperature (°C)
- \( D \) depth (m)
- \( T_s \) sky temperature (K)
- \( g \) gravitational acceleration (m/s²)
- \( t \) time (h)
- \( H \) building Height (m)
- \( u, v, w \) velocity components (m/s)
\( h_{\text{in}}, h_{\text{ex}} \) \( x, y, z \) internal and external CHTC (W/m.K) cartesian coordinates
\( K \) thermal conductivity (W/m.K)
\( k \) kinetic energy of turbulence Greek symbols
\( \rho \) pressure (Pa) \( \alpha \) thermal diffusivity of the fluid (m²/s)
\( \text{Pr} \) Prandtl number (\( \nu/\alpha \)) \( \beta \) coefficient of thermal expansion (K⁻¹)
\( q_s^+, q_L^+, q_L^- \) short-wave heat gain, long-wave heat gain and long-wave heat loss (W/m²)
\( q_s^- \) heat gain and long-wave heat loss (W/m²)
\( \mu_{\text{eff}}, \mu_{\text{mol}}, \mu_{\text{t}} \) effective, molecular and turbulence viscosity (m²/s)
\( \varepsilon \) turbulence energy dissipation rate
\( S \) dimensionless strain rate
\( T \) temperature (K) \( c, r, \tau \) emissivity, reflectivity and transmissivity
\( T_m \) average annual soil temperature (K) \( \rho \) density of the material (kg/m³)
\( T_a, T_{\text{in}} \) atmospheric temperature and indoor temperature (K)
\( \alpha_r, \alpha_e, \alpha_k \) inverse effective Prandtl number for energy, \( \kappa \) and \( \varepsilon \) equations, respectively

REFERENCES