CFD Modeling of Heat Convection in a Large Cross-section Earth-to-Air Heat Exchanger

J. Zhang and F. Haghighat

Department of Building, Civil and Environmental Engineering Concordia University Montreal, Quebec, H3G 1M8 CANADA

ABSTRACT

An Earth-to-Air Heat Exchanger (ETAHE) is a low energy cooling and heating technology for buildings. It uses the ground's thermal storage capacity to dampen ambient air temperature oscillations by delivering outdoor air to indoor through a horizontally buried duct. To reduce the airflow resistance in ETAHEs and save fan energy, some hybrid ventilated buildings have recently adopted large cross-sectional ETAHE ducts, which bring great difficulties for predicting their thermal performance. This is mainly attributed to the complex convective heat transfer between the air and the duct surface. In this study, a computational fluid mechanics (CFD) model of the heat convection is developed using commercial software. A two layer turbulent model is used to resolve detailed flow information in the viscous sub-layer. The model is verified by comparing the simulation results with experiment measurements from literature. The model is further used in a parametric study to investigate the effect of the duct cross-sectional size and the airflow rate on local area-averaged convective heat flux. The model is proved to be a reliable tool to study heat convection in ETAHE.

KEYWORDS

Convective heat transfer, CFD, earth-to-air heat exchanger, two layer turbulent model

INTRODUCTION

An Earth-to-Air Heat Exchanger (ETAHE) ventilates air to the indoor environment through one or several horizontally buried ducts. In this way, the ground's large thermal capacity and relatively stable temperatures are used to preheat or pre-cool the air, resulting in energy savings for the building. Conventional ETAHE systems are installed in mechanically ventilated buildings, in which electrical fans provide the airflow driving forces and the buried pipes are of small diameter. The airflow in these pipes can be considered as fully developed flow. In the last decade, a number of studies had been conducted to simulate the conventional ETAHEs' energy performance. Commonly they divide the duct into a number of control volumes along its running length. A heat balance approach for each control volume, namely heat loss/gain by the air is equal to heat gain/lost by the earth, is adopted to predict the outlet air temperatures. Among those models, some of them (e.g. Tzaferis et al 1992) assume the pipe wall temperature is the same as the undisturbed soil temperature at the same depth; however, others (e.g. Hollmuller 2003) solve dynamic heat conduction and moisture transfer in the surrounding soil in order to obtain more accurate results. Recently, to reduce the airflow resistance in an ETAHE as well as the related fan energy consumption, some hybrid ventilated buildings have adopted very large cross-sectional ducts (Schild 2001). Such systems are very suitable for buildings with displacement ventilation in rooms. The integration of ETAHE and hybrid ventilation is regarded as a new approach to improve building energy efficiency (Heiselberg 2004). But the models developed for small-diameter ducts are not applicable, so new analytical and design tools are required for the large ducts.

CONVECTIVE HEAT TRANSFER IN ETAHE

In 2002, Wachenfeldt conducted an energy simulation to study the overall energy performance of a hybrid ventilated building with a large duct ETAHE. The system's efficiency was demonstrated from the simulation and long term monitoring results. However, the author had to significantly simplify the simulation for ETAHE and to use a very rough convective heat transfer coefficient (CHTC) algorithm developed from measurement. A study by Zhang and Haghighat (2005) showed that the CHTC greatly differs from that of fully developed heat convection in small-diameter pipes. Some observed phenomena in the duct, such as reverse airflow, air temperature stratification, and duct surface temperature variation, indicated the complexity of the heat transfer process. This is attributed to the entrance and buoyancy effects, and the depth differences of the duct surfaces. In building energy simulation, interior surface convection of an enclosure is calculated using Eqn. 1. This indicates interior surface convection, namely the choice of CHTC, can strongly influence the prediction of energy performance.

$$q_w = h_c\left(T_{ref} - T_{surf}\right) \qquad \text{Eqn. 1}$$

In building applications, existing CHTC correlations are mostly developed from experiments. They are usually calculated from controlled enclosure surface temperature, measured reference air temperature and measured convective heat flux. Under steady states, CHTC algorithms can be developed using Eqn. 1. However, a full scale experiment on an ETAHE duct is very difficult. It is even more expensive to study h_c with different design configurations. In addition, the experimental method's accuracy has always been a major challenge in building applications. With the development of CFD, numerical experiments have become a very useful alternative. A few successful studies have proven that CFD can be used for convective heat transfer study, for example Zhai and Chen (2004) and Hsieh (2004). However Zhai and Chen (2004) and Awbi (1998) also pointed out that the result accuracy is sensitive to the turbulent model and the first grid size. Therefore, the objective of this study is to develop a reliable CFD simulation model and use it to study the complex heat convection in ETAHE.

MODEL DESCRIPTION

CFD discretizes a computational domain into a number of grids and solves the governing conservation equations of flow on these grid cells. This study uses a fully-unstructured finite-volume CFD solver, Fluent 6.2. The governing equations are conservation of mass, momentum, and energy. The turbulent model is addressed in the next section. The heat convection in ETAHE is assumed to be quasi-steady state. Air is assumed to be incompressible and the buoyancy effect is accounted for by using the Boussinesq approximation. The SIMPLE algorithm is used for the pressure-velocity coupling in the segregated solver. A second-order upwind scheme is adopted for discretization of the governing equations. The convergence criteria for all variables are set to be 10^{-4} except for energy, which is set to 10^{-6}. In the CFD model the underground duct is represented by a horizontal duct with an inlet tower and an outlet. In practice, different forms of openings can be arranged on inlet tower's walls above the ground surface and the walls are insulated to protect from freezing. Therefore, the ambient air can be assumed to uniformly flow downward in an adiabatic vertical enclosure after entering the tower's inlet openings. Inclusion of the tower in the simulation is only to allow airflow profile to develop before reaching the horizontal duct. In other words, it is to make the inlet boundary condition more realistic. There are different possibilities of designing an airflow path between the end of the ETAHE and the indoor space. Since the ETAHE's ceiling is a couple of meters below the ground, the outlet is located at the ETAHE's ceiling. The duct's symmetric configuration allows defining the simulation domain as half of the duct size to save computational cost. To enable the variation of surface temperature as well as to help analysis of local area-averaged heat transfer, the duct's surface is divided into a number of elements, as shown in Fig. 1.

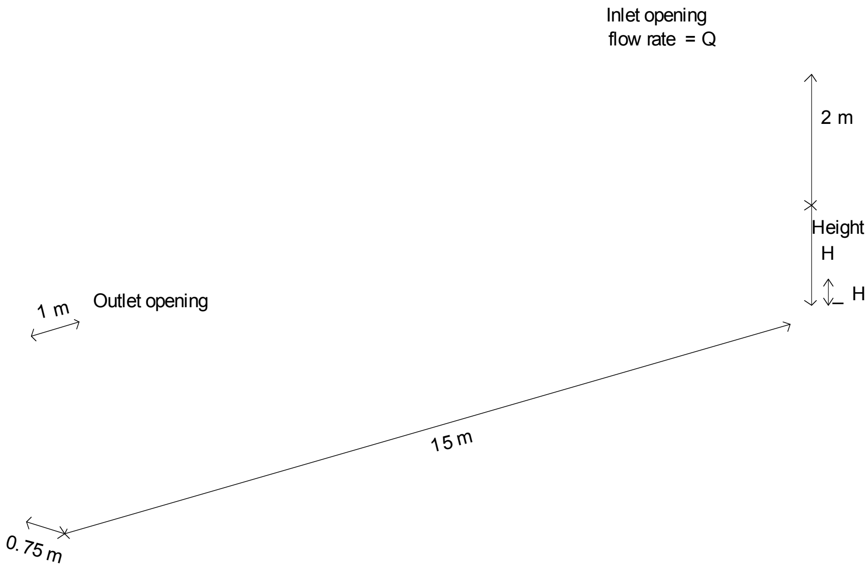

Figure 1. Schematic drawing of the ETAHE model

Two Layer Turbulent Model

In the CFD simulation, surface convective heat flux is defined as Eqn. 2. This indicates that the heat flux is determined by the turbulent viscosity and the temperature gradient at the first grid from the surface. Therefore, to obtain accurate convective heat transfer, the selection of the turbulent model and first grid size are very important. This study uses a two layer turbulent model, which solves a one-equation k–l model (Wolfshtein 1969) in the near-wall region and the standard $k-\varepsilon$ model (Launder and Spalding 1974) in the outer region away from walls. These two regions are separated by a critical turbulent Reynolds number defined in Eqn. 3. For the near-wall (viscous-effected) region, where $\mathrm{Re}_y < 200$, the turbulent viscosity is calculated using Eqn. 4, in which the turbulent length scale l_μ is defined in Eqn. 5.

$$q_w = -\left(\frac{\mu}{\Pr} + \frac{\mu_t}{\sigma_t}\right)\frac{(T_{surf} - T_1)}{D} \quad \text{Eqn. 2}$$

$$\mathrm{Re}_y = \frac{\rho y\sqrt{k}}{\mu} \quad \text{Eqn. 3}$$

$$\mu_t = \rho C_\mu l_\mu \sqrt{k} \quad \text{Eqn. 4}$$

$$l_\mu = y c_l \left(1 - e^{-\mathrm{Re}_y / A_\mu}\right) \quad \text{Eqn. 5}$$

Mesh Development

The two layer turbulent model can resolve detailed turbulent information in the viscous-effected near-wall region, which determines the prediction of convective heat flux. However, this near-wall treatment requires much more computation costs than the standard wall function method, which assumes an empirical velocity profile from the wall to the first grid in the outer region. In the current study, the near-wall region is discretized with a boundary layer mesh, which includes a number of dense grids from the surface. Beside the thickness of the boundary layer mesh, the grid size in the outer region also has important impacts on the simulation's accuracy. The mesh quality in the two regions are separately analysed by conducting grid independence exams. The satisfactory mesh is concluded to have

- uniform hexahedral mesh in the outer region (the size depends on the air flow rate. 0.075 m was verified to be appropriate for simulation cases in this paper)
- 15 grids in the mesh boundary layer arranged using first grid and growth factor approach
- first grid from the surface corresponding $Y^+ \approx 1$
- at least 10 grids in the viscous-effected region where $\mathrm{Re}_y < 200$

Boundary Conditions

At the inlet, uniform velocity distribution normal to the inlet, and the inlet's hydraulic diameter and turbulent intensity are used to determine the turbulent properties. At the outlet, a zero diffusion flux boundary condition for all variables is used. On the horizontal duct surfaces, the temperature of the individual surface element is specified. On the inlet tower surfaces, adiabatic surface condition is applied. In addition, a symmetry boundary condition is used at the symmetry plane of the duct.

MODEL VERIFICATION

According to field measurements conducted by Wachenfeldt (2002), the heat transfer in the ETAHE may be either forced or mixed convection depending on the airflow rate. Therefore, a laboratory experimental study (Spitler 1992) and (Fisher 1995) on mixed and forced convection in a room-size enclosure is used to verify the developed model. The experimental enclosure is shown in Figure 2 (left). The comparison between the measurement results and simulation shown in Figure 2 (right) indicates that the model can predict the heat convection with satisfactory accuracy.

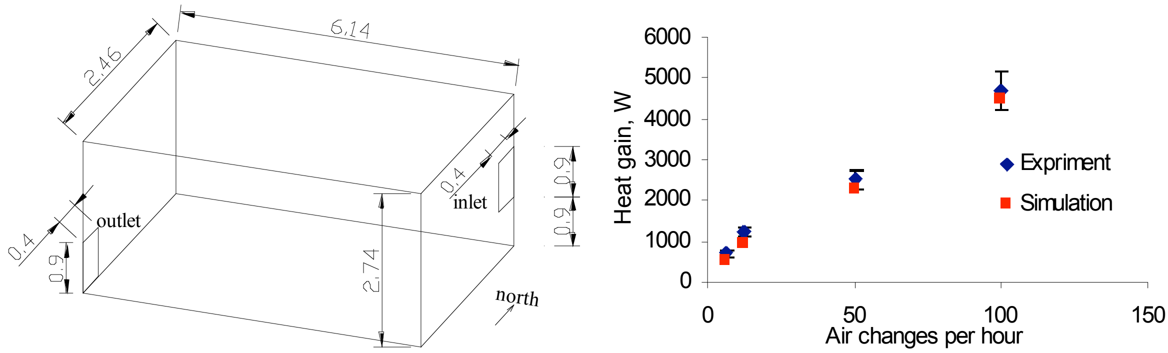

Figure 2. Left: configuration of the experimental chamber
Right: comparison of total heat flux results between measurements and simulations

MODEL APPLICATIONS

The CFD model is used to conduct a parametric study for the effects of air flow rate and duct cross-sectional size on the heat convection. The basic configuration of the duct is shown in Figure 1. All the surfaces are defined to be 10 °C and the inlet air temperature is set to be -10 °C. From Figure 3 (left), one can find that the heat flux is enhanced when the airflow rate increases. In Figure 3 (right), decrease of the cross-sectional size also increases the heat convection. The two factors can cause similar effect on the heat convection because the airflow's momentum depends on both of them. It should be noted the variation of the surface heat flux is very large along the duct length and between the duct ceiling and the floor. Therefore, only if appropriate convective heat transfer coefficients are known, energy simulation could accurately predict the ETAHE's performance.

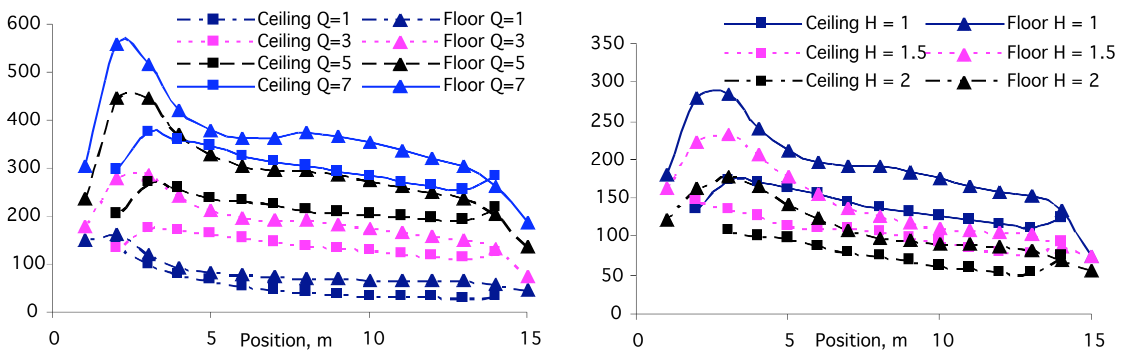

Figure 3. Left: effect of volumetric airflow rate on local area-averaged heat flux with duct height of 1m
Right: effect of duct height on local area-averaged heat flux at airflow rate of 1 m^3/s

CONCLUSION

A numerical simulation model of heat convection in ETAHE is developed using CFD method. In this model, appropriate turbulent model and corresponding mesh generation criteria are determined. The model is verified by comparing a simulation results with an experimental measurement from literature. As applications, the model is used to conduct a parametric study to investigate the effects of airflow rate and duct cross-sectional size on the heat convection. The results showed that large variation of convective heat flux exists at different locations of the duct surface.

NOMENCLATURE

h_c convective heat transfer coefficient, W/(m^2·K)

q_w convective heat flux, W/m^2

T_{ref} reference temperature of the air, °C

T_{surf} surface temperature, °C

Pr Prandtl number

σ_t turbulent Prandtl number for energy

T_1 temperature of air at the first grid next to the wall, °C

μ_t turbulent viscosity kg/(m·s)

μ air viscosity, kg/(m·s)

D the distance of the first grid from the wall, m

Re_y turbulent Reynolds number

ρ air density, kg/m^3

y perpendicular distance of a grid from the nearest wall, m

k turbulent kinetic energy, m^2/m^2

C_μ, c_l, and A_μ constants

l_μ turbulent length scale, m

Y^+ nondimensional parameter

REFERENCES

Awbi, H.B. (1998). Calculation of Convective Heat Transfer Coefficients of Room Surfaces for Natural Convection. *Energy and Buildings* **28: 2**, 219-227.

Fisher, D.E. (1995). *An Experimental Investigation of Mixed Convection Heat Transfer in a Rectangular Enclosure*, Ph.D. Thesis, University of Illinois, USA.

Heiselberg, P. (2004). Building Integrated Ventilation Systems - Modelling and Design Challenges. *CIB 2004 World Building Congress*, Toronto, Canada.

Hollmuller, P. (2003). Analytical Characterization of Amplitude-Dampening and Phase-Shifting in Air/Soil Heat-Exchangers. *International Journal of Heat and Mass Transfer* **46: 22**, 4303-4317.

Hsieh, K.J., and Lien, F.S. (2004). Numerical Modeling of Buoyancy-Driven Turbulent Flows in Enclosures. *International Journal of Heat and Fluid Flow*, **25: 4**, 659-670.

Launder, B.E., and Spalding, D.B. (1974). The Numerical Computation of Turbulent Flows. *Computer Methods in Applied Mechanics and Engineering* **3: 2**, 269-289.

Schild, P.G. (2001). An Overview of Norwegian Buildings with Hybrid Ventilation. *HybVent Forum '01*, Delft University of Technology, The Netherlands.

Spitler, J.D. (1990). *An Experimental Investigation of Air Flow and Convective Heat Transfer in Enclosures Having Large Ventilative Flow Rates*, PhD thesis, University of Illinois at Urbana-Champaign.

Tzaferis, A., Liparakis, D., Santamouris, M., and Argiriou, A. (1992). Analysis of the Accuracy and Sensitivity of Eight Models to Predict the Performance of Earth-to-Air Heat Exchangers. *Energy and Buildings* **18: 1**, 35-43.

Wachenfeldt, B.J. (2003). *Natural Ventilation in Buildings Detailed Prediction of Energy Performance*, PhD Thesis, Norwegian University of Science and Technology.

Wolfshtein, M. (1969). The Velocity and Temperature Distribution in One-Dimensional Flow with Turbulence Augmentation and Pressure Gradient. *International Journal of Heat and Mass Transfer* **12: 3**, 301-318.

Zhai, Z., and (Yan) Chen, Q. (2004). Numerical Determination and Treatment of Convective Heat Transfer Coefficient in the Coupled Building Energy and CFD Simulation. *Building and Environment* **39: 8**, 1001-1009.