

CONVECTIVE HEAT TRANSFER PHENOMENA IN LARGE VENTILATED DUCTS

Jian Zhang, and Fariborz Haghighat [†]

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

ABSTRACT

An Earth-to-Air Heat Exchanger (ETAHE) uses the ground's thermal storage capacity to dampen ambient air temperature oscillations by delivering the outdoor air to the indoors through a horizontally buried duct. Most ETAHE simulation models assume the airflow is hydrodynamically and thermally fully developed at any duct cross sections. They use empirical correlations to calculate convective heat transfer at the duct surface. Recently, to reduce the airflow resistance in ETAHEs and save fan energy, some hybrid ventilated buildings have adopted large cross-sectional ETAHE ducts. This caused great difficulties for predicting their thermal performance because of the complex convective heat transfer process on the duct interior surfaces. In this study, numerical experiments are conducted to investigate the airflow and heat transfer in the large cross-sectional ETAHEs using computational fluid dynamics (CFD) method. A two-layer turbulent model is used to solve detailed flow information in the near-wall region. The model is further used to investigate the effects of duct cross-sectional area and airflow rate on the local convective heat flux.

KEYWORDS

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

INTRODUCTION

An Earth-to-Air Heat Exchanger (ETAHE) is a low energy cooling and heating technology for indoor environment control. It ventilates air to the indoor through one or several horizontally buried ducts, as shown in Figure 1. The ground's massive thermal capacity is used to preheat or pre-cool the air in order to reduce the building's energy consumption.

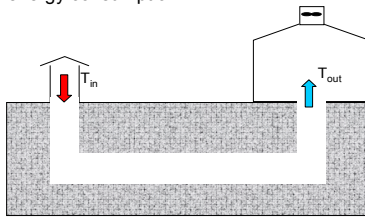


Figure 1. Schematic of ETAHE

In conventional ETAHEs air is delivered through the ducts by mechanical fans. Their hydraulic diameters are two orders of magnitude smaller than their lengths. Therefore, it is assumed that the airflow is hydrodynamically and thermally fully developed. In the past two decades, a number of studies had been conducted to develop simulation models for the energy performance of the ETAHEs. They usually divide a duct into a number of control volumes along its length. The outlet air temperature is predicted based on a heat balance approach for each control volume. The temperature of soil surrounding the buried duct is assumed to be the same as the undisturbed soil temperature. Reviews on this type of models can be found in a comparative study conducted by Tzafiris et al. (1992). The models have lately been improved by taking into account the heat transfer in the ground. The common method is to simultaneously determine the heat conduction in a soil around the ETAHE duct and the heat convection in the duct. Correlations of heat convection for fully developed flow are used to predict the surface heat transfer coefficients. The ground temperature variations due to daily and seasonal

[†] Corresponding Author: Tel: + 1 514 848 2424 ext. 3192, Fax: + 1 514 848 7965
E-mail address: haghi@cbs-engr.concordia.ca

weather changes are assumed to only take place at far-field and at the ETAHE burial depth, and the effect is radically symmetrical.

Recently there has been a new trend in improving energy efficiency of mechanically ventilated buildings. It is based on the reduction of pressure drop in the ductwork by increasing the cross-sectional area in order to reduce fan energy consumption. It is designed to simultaneously or alternatively employ the natural and mechanical airflow driving forces to satisfy the ventilation requirements and it is called hybrid ventilation system (Heiselberg 2002). In the past decade, some ETAHEs have been implemented in hybrid ventilated buildings (Schild 2001). To follow the strategy of the ventilation system, large cross-sectional ducts were chosen to minimize the pressure drop. In this type of application the difference between the duct hydraulic diameters and its length is only one order of magnitude. The change from the conventional small cross-sectional ducts to the large ones appears to be simple. However, the existing simulation models may not be applicable due to the complicated airflow and heat transfer processes. The objective of this study is to numerically investigate the airflow and heat transfer in the large cross-sectional ETAHEs.

CONVECTIVE HEAT TRANSFER IN ETAHE

Although the large cross-sectional ETAHEs have been used for several years, their performance has not been completely studied. Wachenfeldt (2003) investigated the overall energy performance of a hybrid ventilated building with an ETAHE using both simulation technique and long term monitoring. From data analysis and observation he noticed some phenomena, such as air temperature stratification, reverse airflow, and duct surface temperature variations. They were claimed to be the results of a complicated heat transfer process. To obtain convective heat transfer coefficient (CHTC) inputs for his building energy simulation, limited amount of data (i.e. the temperatures of inlet air, outlet air, and duct surfaces) was collected. Zhang and Haghghat (2005) showed that the CHTC of a large duct could greatly differ from that of fully developed heat convection in small-diameter pipes. This is due to entrance effects, buoyancy effects, and depth differences between the duct roof and floor. When an energy simulation is conducted, the interior surface convection of an enclosure is usually calculated using Equation 1.

$$q_w = h_c A (T_{ref} - T_{surf}) \quad \text{Equation 1}$$

For room applications, interior surface CHTCs are usually determined by empirical correlations generalized from experimental data. Test chambers need to be built to simulate the practical heat convection. If the similar approach is taken to develop a correlation for ETAHEs, a full scale duct model needs to be constructed. This is very expensive in terms of its construction and operation. It is even more laborious to run the experiments when design configurations need to be changed, such as duct length, width, height, and inlet/outlet forms. In addition, the accuracy of the experimental method has always been a major challenge in building applications (Spitler 1990 and Fisher 1995). With the development of CFD, numerical experiments become an alternative way to study heat convection. A few successful studies have been presented by Zhai and Chen (2004) and Hsieh (2004). The flexibility of numerical experiments in defining boundary conditions and design configurations overcomes many difficulties encountered in laboratory experiments. Zhai and Chen (2004) and Awbi (1998) also revealed the shortcoming of this approach, which is the sensitivity of the simulated heat convection to the turbulent model and the first grid size. Therefore, a reliable CFD modeling method needs to be determined.

CFD MODEL DESCRIPTION

Computational fluid dynamics technique discretizes a simulation domain into a number of grids and solves the governing conservation equations of flow and energy on these grids. The solution includes detailed information of the flow variables at every grid point. In the current study, the computational domain is the air volume encompassed by solid surfaces and inlet/outlet openings. The ETAHE is represented by a horizontal duct with an inlet tower and an outlet as shown in Figure 2. The symmetric configuration of the ETAHE allows cutting the simulation domain to its half to save computational time. To enable the analysis of heat transfer rate at various locations, the duct surfaces are divided into a number of elements.

Governing equations

Airflow in building ventilation applications is mostly turbulent and incompressible. The governing equations for mass, momentum, and energy in Reynolds-averaged forms can be simplified using the eddy viscosity concept as shown in Equation 2, 3, and 4. The buoyancy force caused by density gradients in the fluid is taken into account using the Boussinesq approximation.

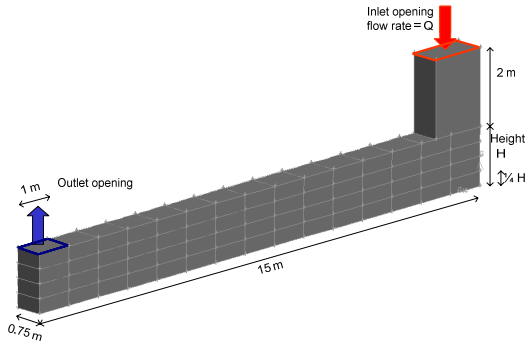


Figure 2. Schematic drawing of the ETAHE model

$$\frac{\partial u_j}{\partial x_i} = 0 \quad \text{Equation 2}$$

$$\frac{\partial}{\partial x_i} (\rho c_p u_i T) = \frac{\partial}{\partial x_i} \left[\left(\lambda + \frac{c_p \mu_t}{\sigma_t} \right) \frac{\partial T}{\partial x_i} \right] \quad \text{Equation 3}$$

$$\frac{\partial}{\partial x_i} (\rho u_i u_i) = -\frac{\partial \bar{P}}{\partial x_j} + \frac{\partial}{\partial x_i} \left(\mu_t + \mu \left(\frac{\partial u_j}{\partial x_i} + \frac{\partial u_i}{\partial x_j} \right) \right) + \rho \beta (T_0 - T) g_j \quad \text{Equation 4}$$

Turbulent modeling

In a CFD simulation, convective heat flux in turbulent flow is mathematically expressed as Equation 5.

$$q'_w = -\left(\frac{\mu}{Pr} + \frac{\mu_t}{\sigma_t} \right) \frac{(T_{surf} - T_i)}{D} \quad \text{Equation 5}$$

This indicates that the critical variables affecting the heat flux prediction are the turbulent viscosity and the temperature gradient from the first grid to the surface. It also explains why turbulent model and first grid size are extremely critical to the heat convection prediction. In room airflow simulations, the standard $k-\varepsilon$ turbulence model with log-law wall functions has been used the most often. Although many successful studies have been carried out, the model has been shown deficient in predicting surface convective heat transfer due to two reasons. First, the standard $k-\varepsilon$ model was formulated for fully developed turbulent flows and it over-predicts the eddy viscosity in low velocity regions. Second, the log-law functions require the first grid to be out of the viscous sub-layer, within which the temperature gradient is important for convection simulation. Some improvements have been found when Low-Reynolds number models were used for room airflow simulation (Chen et al. 1990, Haghghat et al. 1992 and Awbi 1998). However, the computational expense for full scale room simulation is considerably high. Therefore, improving the near-wall treatments for the standard $k-\varepsilon$ model becomes an opportunity. 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. The demarcation of the two regions is determined by a critical turbulent Reynolds number defined in Equation 6. For the outer region, $Re_y > 200$, the standard $k-\varepsilon$ model is used. For the viscous-affected near-wall region, $Re_y < 200$, the $k-l$ model is used. The momentum equations and the k equation are retained the same as in standard $k-\varepsilon$ model. The turbulent viscosity, μ_t , is computed using Equation 7, in which the turbulent length scale l_μ is defined by Equation 8 (Chen and Patel 1988). To smoothly blend the two regions, an enhanced wall treatment method is adopted (Fluent Inc. 2003).

$$Re_y = \frac{\rho y \sqrt{k}}{\mu} \quad \text{Equation 6}$$

$$\mu_t = \rho C_\mu l_\mu \sqrt{k} \quad \text{Equation 7}$$

$$l_\mu = y c_l \left(1 - e^{-Re_y / A_\mu} \right) \quad \text{Equation 8}$$

Mesh development

To take advantage of the two-layer turbulent model, the mesh of computational domain has to be developed to form two adjacent regions, i.e. a near-wall region and an outer region. In this study, hexahedral elements are generated throughout the computational domain using commercial grid generation software, Gambit 2.3. A fine mesh is constructed in the near-wall region. Based on the first grid size, i.e. the distance of the wall to its adjacent grid, the grids proportionally grow away from the wall. In the turbulent region, hexahedral elements are generated with a uniform characteristic length. Detailed mesh generation criteria have been established in the previous study by Zhang and Haghghat (2006).

Boundary conditions

The inlet tower's walls are normally insulated to protect them from freezing. Therefore, the walls above the duct ceiling are set to be adiabatic. Inclusion of the tower in the computational domain is only to allow the airflow to develop before reaching the horizontal part. Since the tower is usually a few meters high from the duct ceiling, a uniform velocity boundary condition can reasonably be applied at the inlet opening. At solid surfaces, no-slip condition:

$$u_j = 0 \quad \text{Equation 9}$$

$$T = T_{surf} \quad \text{Equation 10}$$

A zero diffusion flux for all flow variables in the direction normal to the exit opening is applied at the outlet. Turbulent parameters at the inlet are defined using turbulent intensity and inlet characteristic length (hydraulic diameter) method.

$$k = \frac{3}{2} (u_{avg} I)^2 \quad \text{Equation 11}$$

$$\varepsilon = C_\mu^{3/4} \frac{k^{3/2}}{l} \quad \text{Equation 12}$$

Verification of simulation results

A laboratory experimental study (Spitler 1992 and Fisher 1995) on mixed and forced convection in a room-size enclosure was conducted to develop correlations of convective heat transfer coefficients. Detailed measurements were made especially on quantifying the surface heat flux. Therefore, the study was selected to verify the CFD model. The experimental enclosure is shown in Figure 3 (left). The comparison between the measurement results and simulation shown in Figure 3 (right) indicates that the model can predict the heat convection with satisfactory accuracy.

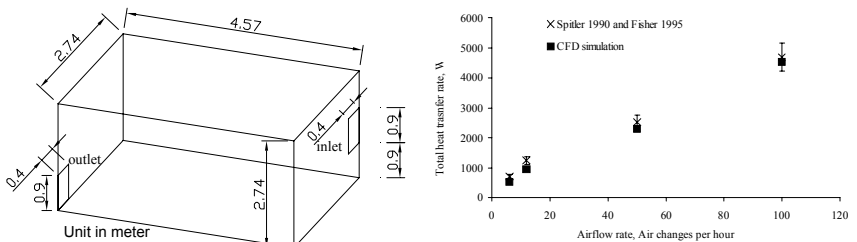


Figure 3. Left: configuration of the experimental chamber

Right: comparison of total heat flux results between measurements and simulations

PARAMETRIC STUDY

The CFD model is used to study the effects of two design parameters on the heat convection: air flow rate and duct height. The configuration of the duct is shown in Figure 2. The isothermal temperature of the duct surface is 10°C and the inlet air temperature is -10°C. Various airflow rates, i.e. 1 m³/s, 3 m³/s, 5 m³/s, and 7 m³/s, were selected to simulate different ventilation requirements of the system. The area-averaged heat flux over each surface element is calculated. The simulated surface heat flux results at different surface elements of duct ceiling and floor are plotted in Figure 4. By comparing the heat flux results from cases with different airflow rates, one can find that the heat convection was enhanced when increasing the airflow rate.

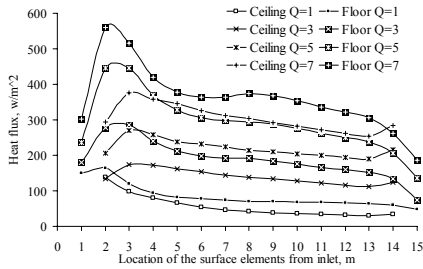


Figure 4. Effect of volumetric airflow rate on local area-averaged heat flux with duct height of 1 m

The effects of duct height are evaluated by three simulation cases with different heights, i.e. 1 m, 1.5 m, and 2 m. The duct surface temperature is 10°C. The inlet air temperature is -10°C. The airflow rate is 3 m³/s. The results are plotted in Figure 5. It shows that decrease of the cross-sectional size increases the heat convection.

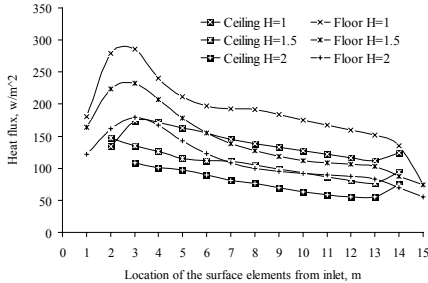


Figure 5. Effect of duct height on local area-averaged heat flux at airflow rate of 1 m³/s

In Figures 4 and 5, a bump can be found at the beginning of every floor curve. This is mainly attributed to the rectangular turn at the inlet tower's bottom, which directs the air to flush over the duct floor. Accordingly, a stagnant flow region is formed at the bottom corner of the duct. This causes a relatively low heat flux at the first element of the surface. In addition, it is noted that the difference of the surface heat flux between the ceiling and the floor is very large. Beside the air-flushing effects, buoyancy-driven airflow is believed to be the main reason for the difference.

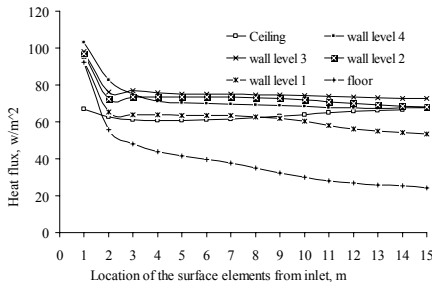


Figure 6. Local area-averaged heat flux at different surfaces of the duct

Further analysis on buoyancy effects

To investigate the effects of the buoyancy-driven airflow on the heat convection, another simulation was conducted. In order to eliminate the air flushing effects, the ETAHE's configuration was changed to a simple rectangular duct with height of 1 m. Its inlet and outlet openings become perpendicular to the duct length, and they are located at the two ends of the duct. The airflow rate is $1 \text{ m}^3/\text{s}$. The temperature of the duct surfaces is 10°C and the inlet air is 30°C . The area-averaged heat flux results at different surface locations are shown in Figure 6. It is noted that except for the entrance region the heat flux curves are relative flat. However, the differences between the floor and other surfaces are still very large although uniform velocity boundary condition is used at the inlet. To find the reason, the air temperature profiles at different vertical axes on the middle plane are plotted in Figure 7.

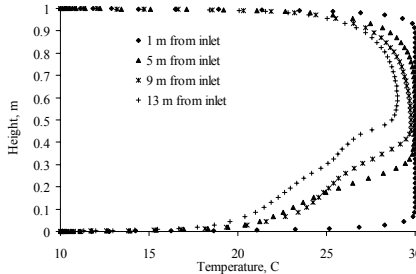


Figure 7. Development of air temperature profiles at the middle plane of the duct

As shown in Figure 7, the warm air from the inlet is gradually cooled along the duct length. The cooled air tends to move to the duct bottom. The development of the profiles along the flow indicates that the buoyancy force caused vertical temperature stratification in the duct. As results, the air temperature gradients near the floor become much less than those near the ceiling.

CONCLUSION

Field observation and earlier investigations have shown that simple change of ETAHE duct size can cause great difficulties in simulating their energy performance. In the large cross-sectional duct, the airflow and heat transfer are significantly different from thermally and hydrodynamically fully developed flow. Since the existing heat convection correlations are not applicable for such ETAHE problem, this research was aimed to use CFD modeling method to study the heat convection in the ducts. Knowing the deficiency of the standard $k - \varepsilon$ turbulent model with log-law wall function, a two-layer turbulent modeling method was selected. The comparison of results from simulations and experiments showed that the CFD method can simulate the heat convection process with satisfactory accuracy. The model was used to investigate the effects of airflow rate and duct height on the heat convection. As expected, increasing the airflow momentum by either increasing the airflow rate or decreasing the cross-sectional area enhanced the heat convection when other design parameters are fixed. It is important to notice that the variation of the surface heat flux is very large along the duct length and between the duct ceiling and the floor. To independently investigate the buoyancy effects on the heat transfer, the further simulation was performed on a simple horizontal duct. It was found that although uniform air velocity was applied at the inlet the differences of heat flux between the ceiling and the floor are still very large. Buoyancy force is identified to be the reason for the difference. The analysis indicates that an appropriate convective heat transfer model needs to be developed from future numerical experiments in order to accurately design and simulate large cross-sectional ETAHEs.

NOMENCLATURE

- A surface area, m^2
- A_μ constant
- c_f constant
- c_p specific heat capacity, $\text{J}/(\text{kg}\cdot\text{K})$
- C_μ constant

D distance of the first grid from the wall, m
 g_j gravity acceleration in x_j direction, m/s^2
 h_c convective heat transfer coefficient, $W/(m^2 \cdot K)$
 I turbulent intensity
 k turbulent kinetic energy, m^2/m^2
 l turbulence length scale, m
 l_μ turbulent length scale, m
 Pr Prandtl number
 \bar{P} effective pressure, pa
 q_w convective heat flow rate, W
 q'_w convective heat flux, W/m^2
 Re_s turbulent Reynolds number
 T temperature, $^\circ C$
 T_0 reference temperature, $^\circ C$
 T_1 temperature of air at the first grid next to the wall, $^\circ C$
 T_m inlet air temperature, $^\circ C$
 T_{out} outlet air temperature, $^\circ C$
 T_{ref} reference temperature of the air, $^\circ C$
 T_{surf} surface temperature, $^\circ C$
 u velocity, m/s
 u_{avg} average velocity, m/s
 u_i and u_j are velocity in x_i and x_j directions, respectively, m/s
 u_T friction velocity, m/s
 x_i and x_j (i and $j = 1, 2, 3$) three perpendicular coordinates
 y perpendicular distance of a grid from the nearest wall, m
 β thermal expansion coefficient of air
 ε turbulent kinetic energy dissipation rate
 λ thermal conductivity, $W/(m \cdot K)$
 μ physical viscosity, $kg/(m \cdot s)$
 μ_t turbulent viscosity, $kg/(m \cdot s)$
 ρ density, kg/m^3
 σ_t turbulent Prandtl number for energy

REFERENCES

1. A. Tzaferis et al. (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 pp 35-43.
2. P. Heiselberg (ed.) (2002) "Principle of Hybrid Ventilation", Hybrid Ventilation Center, Aarhus University, Denmark.
3. P. G. Schild (2001) "An Overview of Norwegian Buildings with Hybrid Ventilation", HybVent Forum '01, Delft University of Technology, The Netherlands.
4. B. J. Wachenfeldt (2003) "Natural Ventilation in Buildings Detailed Prediction of Energy Performance", PhD Thesis, Norwegian University of Science and Technology.
5. J. Zhang and F. Haghighat (2005) "Simulation of Earth-to-Air Heat Exchangers in Hybrid Ventilation Systems", Proceedings of the Ninth International IBPSA Conference, Building Simulation 2005, Montreal, Canada, 3 pp 1417-1424, Aug. 15-18, 2005.
6. J. D. Spitler (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.
7. D. E. Fisher (1995) "An Experimental Investigation of Mixed Convection Heat Transfer in a Rectangular Enclosure", Ph.D. Thesis, University of Illinois, USA.

8. Z. Zhai and Q. Chen (2004) "Numerical Determination and Treatment of Convective Heat Transfer Coefficient in the Coupled Building Energy and CFD Simulation", *Building and Environment* 39: 8 pp 1001-1009.
9. K. J. Hsieh, and F. S. Lien (2004) "Numerical Modeling of Buoyancy-Driven Turbulent Flows in Enclosures", *International Journal of Heat and Fluid Flow* 25: 4 pp 659-670.
10. H. B. Awbi (1998) "Calculation of Convective Heat Transfer Coefficients of Room Surfaces for Natural Convection", *Energy and Buildings* 28(2) pp 219-227.
11. Q. Chen et al. (1990) "Prediction of Buoyant, Turbulent Flow by a Low Reynolds-Number $k - \epsilon$ Model", *ASHRAE Transactions*, 96: 1 pp 564-573.
12. F. Haghighat (1992) "Air Movement in Buildings Using Computational Fluid Dynamics", *The ASME Journal of Solar Energy Engineering*, 114: 2, pp. 84-92.
13. M. Wolfshtein (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 pp 301-318.
14. Fluent Inc. (2003) "Fluent 6.1 user's guide".
15. B. E. Launder and D. B. Spalding (1974) "The Numerical Computation of Turbulent Flows", *Computer Methods in Applied Mechanics and Engineering* 3: 2 pp 269-289.
16. H. C. Chen and V. C. Patel (1988) "Near-Wall Turbulence Models for Complex Flows Including Separation", *AIAA Journal*, 26(6) pp 641-648.
17. J. Zhang and F. Haghighat (2006). "CFD Modeling of Heat Convection in a Large Cross-Section Earth-to-Air Heat Exchanger", AIVC 27th conference - EPIC2006AIVC, Technologies & sustainable policies for a radical decrease of the energy consumption in buildings, Lyon, France, 3, pp 607-612, 20-22 November 2006.