CFD Simulation and Experimental Data for a Fixed Heat Load Natural Draft Air-Cooled Heat Exchanger with Cold Inflow Mitigation

Christopher C.M. Chu, Md. Mizanur Rahman and Sivakumar Kumaresan

Thermal and Environmental Research Group, Material and Mineral Research Unit, Faculty of Engineering, Universiti Malaysia Sabah Kota Kinabalu, Sabah, MALAYSIA
E-mail: chrischu@ums.edu.my
Tel: +6088 320000 Ext. 3135 Fax: +6088 320348

ABSTRACT
CFD simulation was carried out to corroborate experimental data at fixed heat load of nominally 2.3kW from a natural draft heat exchanger of face dimensions of 0.75m × 0.75m, with or without mitigation of the cold inflow at the chimney exit, where mitigation by installing wire mesh on top of the chimney has been shown by the experiments to enhance air flow rate. A chimney model was simulated at fixed heat loads in a still surrounding at ambient temperature of 30 degree Celsius and atmospheric pressure for two modes: Mode 1 and Mode 0 for with and without a flow resistor (wire mesh) respectively at the top exit. It was found that the simulation could reproduce most of the trends of the experimental data, but had a tendency to magnify the detrimental effects of cold inflow and exaggerate the remedial action of wire mesh in preventing cold inflow, as reflected by the ratio of Mode 1 to Mode 0 air mass flowrate by a factor of up to 2.36, compared to 1.50 in the experimental data. In both simulation and experiment, the average air flow rates obtained at chimney heights of 0.35m, 0.65m, 0.95m and 1.25m, showed progressive increase of air mass flow rate for all cases. Both experimental and simulated heat gain in Mode 0 were more or less constant until the highest chimney height where they showed apparent breakout upwards, whereas in Mode 1 the experimental heat gains gently reduced to a plateau while the simulated heat gains hovered at around 2.3kW. The back-calculated values of Mode 0 experimental outlet temperature at between 140 to 240°C raises concern of hotspot in some electronic components by the ineffectiveness of chimney systems without cold inflow mitigation. Further experiments of similar scale with steadier control of heat flux and heating temperature, and simulating with other turbulence models in transient mode will improve understanding in both Mode 0 and Mode 1 of operation.

INTRODUCTION
In a series of experiments carried out, involving a chimney research facility with geometrical dimensions of 0.75 × 0.75m², 1.00 × 1.00 m² and 1.50 × 1.50 m² face dimensions of a natural convection air-cooled heat exchanger operating at fixed heat loads, results have shown that there is significant difference between the values of the velocity and temperature parameters of a modified chimney and an unmodified, conventional chimney (Rahman, 2011; Chu et al, 2012a; Chu et al, 2012b; Chu and Rahman, 2009) due to cold inflow phenomenon (Jörg, O. and Scorer, 1967; Modi and Torrance, 1987). An important claim in the research finding was that a wire mesh placed on the top exit of a chimney, covering the entire cross-sectional flow area, acted as a cold inflow preventer, which consequently improved the draft over the system with conventional chimney that encounters cold inflow. This is counter-intuitive and has never been reported in the literature on solar chimney and cooling towers as far as the authors are aware (Goodarzi. and Kaimanesh, 2013; Khanal and Lei, 2014; Khanal and Lei, 2012; Kröger, 2004; Zhou et al, 2010). A simulation of a simple cylindrical chimney operating under natural convection has shown that the use of a flow resistor like a wire mesh at the chimney top exit would mitigate cold inflow and consequently enhances the air flow rate and heat discharge rate (Chu et al, 2014). The purpose of this paper is to simulate the Rahman (2011) experimental data, particularly the enhancement of flow rate using wire mesh for natural draft chimneys suffering from cold inflow, by using commercial CFD software, and report on the result of simulation.
**METHODOLOGY**

The scope of the study was limited to the 0.75 × 0.75m² heat exchanger, at chimney heights of 0.35m, 0.65m, 0.95m and 1.25m. The schematic diagram of the CFD set up (Figure 1a) was to have similar geometrical, heat transfer and flow characteristics as that described in Chu et al (2012b), beginning from the square duct housing the electrical coil heater, which delivered heating at 2.30-2.44kW. Two modes of operation were run: Mode 0 for no wire mesh at the top exit and Mode 1 for with wire mesh at the top exit. The system was truncated to the part from the heater downstream, and the upstream part was represented by an artificial layer of resistance exhibiting the same pressure drop characteristics as the facility. The pressure drop upstream of the heater, including inlet contraction, anemometer, bends and divergence, is represented by the equations given in Chu et al (2012b), obtained by curve-fitting all the pressure drop data upstream of the coil heater during the experiments:

Mode 0 \[ \Delta p_U = 0.8065u_i^{1.823} \] (1)
and for Mode 1 \[ \Delta p_U = 0.7604u_i^{1.8586} \] (2)

The pressure drop equation across a single layer of wire mesh is given by (Chu, 1986):

\[ \Delta p_w = 4.42u_S^{1.361} \] (3)

which was obtained by curve-fitting isothermal pressure drop data for velocities 0 to 0.5ms⁻¹. The wire mesh was modeled as a porous plate in the simulation.

The ratio of pipe inlet velocity \( u_i \) to superficial velocity \( u_S \) through the duct area is 64.96. The coefficient of determination of the regression fit to equations (1) and (2) to data was 0.92, and equation (3) has an estimated error of ±5 per cent.

![Figure 1a: The Simulated Chimney System](image)

The CFD software employed was Phoenics 2014 (CHAM UK), which uses SIMPLEST to solve the single-phase finite volume equations of conservation of mass, momentum and energy (CHAM) across the grid mesh shown in Figure 1b:

\[
\frac{\partial (\rho \phi)}{\partial t} + \frac{\partial}{\partial x} \left( \rho U \phi - \Gamma_{\phi} \frac{\partial \phi}{\partial x} \right) = S_{\phi}
\]

where \( \phi \) = the variable in question
\( \rho \) = density
\( U \) = vector velocity
\( \Gamma_{\phi} \) = the diffusive exchange coefficient for \( \phi \)
\( S_{\phi} \) = the source term

Particular forms are:

**Momentum** \( \phi = u, v, w \)

\[ \Gamma_{\phi} = \rho \left( v_i + v_f \right) \] (3)

\[ S_{\phi} = -\frac{\partial p}{\partial x} + \text{gravity + friction + . . . .} \] (4)

**Enthalpy** \( \phi = h \)

\[ \Gamma_{\phi} = \rho \left( \frac{v_i}{Pr_i} + \frac{v_f}{Pr_f} \right) \] (6)

\[ S_{\phi} = -\frac{Dp}{Dt} + \text{heat sources + . . . .} \] (7)

**Continuity** \( \phi = 1 \) (8)
where \( v_t \), \( v_l \) are the turbulent and laminar viscosities, and \( \Pr_l, \Pr_t \) are the turbulent and laminar Prandtl numbers.

The default and recommended turbulence model is the Chen-Kim’s (1987) modified \( \kappa-\varepsilon \) model with the following settings: \( \sigma_\kappa \), coefficient for turbulent kinetic energy \( \kappa = 0.75; \)

\( \sigma_\varepsilon \), coefficient for its dissipation rate \( \varepsilon = 1.15; \) model constants \( C_1 = 1.15, C_2 = 1.9 \) and \( C_3 = 0.25. \)

The ambient condition was atmospheric pressure at 101.325 kPa, 30°C without prevailing wind. Grid mesh sizes were chosen in proportion to the dimensions so as to ensure acceptable levels of accuracy. The simulation was run at steady state with a global convergence criterion of 1 per cent. For Mode 0 however, convergence could not be achieved which can be indicated by the erratic difference between the inlet and outlet velocities, in line with the caution from Andreozzi et al (2009) that cold inflow is not a steady state phenomenon and could only be ‘averaged’ at best by simulation. Area-averaged values of mass flow rate, velocity, and temperature were taken at the inlet and outlet of the chimney system. It should be noted that the outlet temperature readings in the experiments were read at the centres of quadrants and at the centre of the square duct flow area, five in all, and located half-way up the chimney, i.e. at 17.5cm, 32.5cm, 47.5cm and 62.5cm for chimney heights of 35cm, 65cm, 95cm and 125cm respectively above the heating coil. The standard deviation of the type-K thermocouple readings was 6K between the outlet thermocouples (Chu et al, 2012b), which would be erratic for Mode 0 experiments with smaller temperature rise due to cold inflow but not significant in Mode 1. The error band for flowrate measurement was likely to be insignificant in its contribution to heat gain despite the anemometer operating at the low-end 5 per cent of its full range of 0 – 30 ms\(^{-1}\), with a typical \( \pm \)1 per cent error of a reading. However, \( \pm \)5 per cent was assumed in the analysis here.

The grid convergence was determined by observing the simulated mean air mass flowrate, average outlet temperature and heat gain. Based on the chimney configuration of 35 cm high × 75 cm long × 75 cm wide, it can be seen in Table 1 that the grid size of X- and Y-planes at 23 cells, 9 cells, 15 cells in regions 1, 2 and 3 respectively, and Z-plane at 8, 2, 9, 19 cells in regions 1 to 4 counting from the bottom of the domain appears adequate in Mode 1 with a maximum error of 1.5 per cent in mean air mass flow rate, 1.8 per cent in temperature rise and 0.9 per cent in heat gain, between the coarse and the fine grids. In Mode 0 where the flow in experiments and simulation was shown to be unsteady, the differences were significantly larger at 7 per cent, 10 per cent and 20 per cent respectively.

**Table 1: Grid Convergence Analysis**

<table>
<thead>
<tr>
<th>Number of Grid Cell X- &amp; Y-plane</th>
<th>Z-plane</th>
<th>Air Mass Flowrate (kg/s)</th>
<th>Outlet Temp. (°C)</th>
<th>Heat Gain (W)</th>
</tr>
</thead>
<tbody>
<tr>
<td>35 cm × 75 cm × 75 cm</td>
<td>Q = 2320 W</td>
<td>Mode 0</td>
<td>(Without Screen at Exit)</td>
<td></td>
</tr>
<tr>
<td>23, 9, 15</td>
<td>8, 2, 9, 19</td>
<td>6.00 × 10(^3)</td>
<td>44.7</td>
<td>88.9</td>
</tr>
<tr>
<td>28, 18, 30</td>
<td>8, 2, 9, 19</td>
<td>5.84 × 10(^3)</td>
<td>43.4</td>
<td>78.8</td>
</tr>
<tr>
<td>56, 36, 60</td>
<td>8, 2, 9, 19</td>
<td>6.24 × 10(^3)</td>
<td>44.9</td>
<td>94.2</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Number of Grid Cell X- &amp; Y-plane</th>
<th>Z-plane</th>
<th>Air Mass Flowrate (kg/s)</th>
<th>Outlet Temp. (°C)</th>
<th>Heat Gain (kW)</th>
</tr>
</thead>
<tbody>
<tr>
<td>35 cm × 75 cm × 75 cm</td>
<td>Q = 2330 W</td>
<td>Mode 1</td>
<td>(With Screen at Exit)</td>
<td></td>
</tr>
<tr>
<td>23, 9, 15</td>
<td>8, 2, 9, 19</td>
<td>1.34 × 10(^2)</td>
<td>203.1</td>
<td>2.35</td>
</tr>
<tr>
<td>28, 18, 30</td>
<td>8, 2, 9, 19</td>
<td>1.36 × 10(^2)</td>
<td>200.0</td>
<td>2.33</td>
</tr>
<tr>
<td>56, 36, 60</td>
<td>8, 2, 9, 19</td>
<td>1.36 × 10(^2)</td>
<td>200.0</td>
<td>2.33</td>
</tr>
</tbody>
</table>

**RESULTS AND DISCUSSION**

Comparison was made between experiments and simulation by examining the air mass flow rates, temperature rise and heat gains in Mode 0 and in Mode 1, as shown in Figures 2, 3 and 4. The trends were generally in agreement and the closest appears to be the air mass flow rates in Mode 1 at all chimney heights, where the simulation under-predicted the flowrate at lowest chimney height, and over-predicted at higher chimney height (Figure 2). However, the worst agreement appears to be the heat gain also in Mode 1. The simulated heat gain appears to be holding at the fixed heat load but the experimental heat gain slowly reduced asymptotically. Despite the difficulty in convergence in Mode 0, by referring to the vertical air velocity at the monitoring probe, \( \pm \)10 per cent at best, simulated temperature rise for all chimney heights had the right order of magnitude as the experimental values, ranging from \( \pm \)3.22 to 6.45 K, or \( \pm \)17.6 to 31 per cent (Figure 3). They also exhibited a similar trend of a minima in relation to chimney height. Two sources of error were identified: one was the unstable flow fluctuations mentioned above due to the cold inflow in Mode 0; and the other arose from temperature measurement using the area-averaged method in the experiments. It was found that the Favre-averaged values of heat gain of the simulation of Mode 0 experiments were much closer to the supplied heat load. Much of the heat gain error can be attributed to temperature measurement. To make the comparison of heat gain on the same basis area-averaged values only were used in both experiments and simulation. Heat gain in Mode 1 for both experiments and simulation approached that of the supplied amount (Figure 4b), while those of Mode 0 were far below the supplied amount (Figure 4a). To provide a perspective of the extent of the deviation the percentage of heat gain are plotted in Figures 4c and 4d, which show that the excessive deviation between experimental and simulated heat gains from the heat supplied by more than 80 percent could be rectified by using Favre-averaging to within \( \pm \)5 per cent, and that the agreement between experimental and simulated heat gains is reasonable. It is extremely difficult in practice to use
Favr-averaging for temperature measurement and therefore by resorting to area-averaging the deviation could be misinterpreted to heat loss to surroundings.

The CFD was performing conservatively in Mode 0 in that the air mass flowrate, temperature rise and heat gain all were predicted below the experimental values. On the other hand, in Mode 1 the CFD tended to over-predict significantly, with the exception of air mass flowrate. Assuming that the artificial resistance has been correctly applied, the implication of these results is that with cold inflow occurring, the CFD tends to magnify the problem, but it also tends to exaggerate the remedial effect of wire mesh in preventing cold inflow. This is reflected in the enhancement of air mass flow rate in Mode 1 over Mode 0 in the simulation to be in the range of 2.23 to 2.36 for all the chimney heights, while in the experiments it was found to be 1.45 to 1.50 (Chu and Rahman, 2009).

A possibility that should not be ignored is the maximum surface temperature of the heater under Mode 0. Both the experimental and simulated values of the outlet temperatures using area-averaging showed relatively low temperature with respect to the ambient. The Favre-averaged heat gain by the air implies the medium was removing heat from the heater at the cost of a much higher temperature. If the outlet temperature is back-calculated by using the heat supplied however, the value could reach between 140°C at the highest chimney height of 1.25 m to 240°C at the lowest chimney height of 0.35 m. This potential hotspot can be a cause of concern if natural convection chimney is contemplated to cool electronic components which have heat constantly generated by electrical means.

The experimental values were collected under difficult conditions as the Froude number was well below 1.00 and the duct size meant the flow might not be controlled to the ideal precision with some parts of the wire mesh sagging somewhat. It is recommended that another chimney facility capable of measuring low velocity non-intrusively and work in either fixed heat flux or fixed temperature be designed and built for gathering more data. In the simulation, other turbulence models might be used for comparison and in transient mode to strengthen these findings.

Figure 2: Trend of air mass flowrates with chimney height at fixed heat load

Figure 3a: Temperature rise in relation with chimney height at fixed heat load in Mode 0

Figure 3b: Temperature rise in relation with chimney height at fixed heat load in Mode 1

Figure 4a: Heat gain with chimney height at fixed heat load in Mode 0
CONCLUSIONS
Comparison between experimental and simulation data of a 0.75x0.75m² duct with chimney heights of 0.35m, 0.65m, 0.95m and 1.25m operating under natural convection demonstrates similar trends in air mass flow rate, temperature rise and heat gain, where the enhancement of air mass flow rate found in experiments by placing wire mesh at the top exit of a chimney encountering cold inflow was corroborated by simulation.
While the CFD as currently set up could simulate the trends, it tended to magnify the problem of cold inflow in deteriorating the draft and to exaggerate the remedial effect of wire mesh in restoring the draft.

ACKNOWLEDGMENT
The authors would like to express their sincere thanks to the Ministry of Education, Malaysia, for providing funding of fundamental research under grant No. FRG0022-TK-1/2006 and FRG0197-SG-1/2010.

NOMENCLATURE

\( a, b \) coefficients of pressure drop equation \\
\( k \) phase number, 1 or 2 in Phoenics. \\
\( Pr_t, Pr_l \) the turbulent and laminar Prandtl numbers. \\
\( S_\phi \) the source term in CFD calculation \\
\( t \) time \\
\( U \) vector velocity in CFD calculation \\
\( u, v, w \) \( x-, y- \) and \( z- \) components of velocity vector \\
\( u_i \) pipe inlet velocity in experimental facility \\
\( u_s \) superficial vertical velocity of duct \\
\( x_k \) distance in \( k \)-phase \\
\( \Delta p_U \) pressure drop upstream of heater \\
\( \Delta p_w \) pressure drop across wire mesh \\
\( \Gamma_\phi \) the diffusive exchange coefficient for \( \phi \) \\
\( \phi \) the variable in question in CFD calculation \\
\( \rho \) density \\
\( \nu_t, \nu_l \) the turbulent and laminar viscosities

REFERENCES


